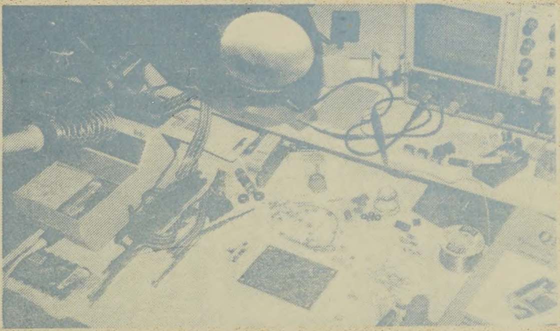
Vol. VI, No. 3 Autumn 1998

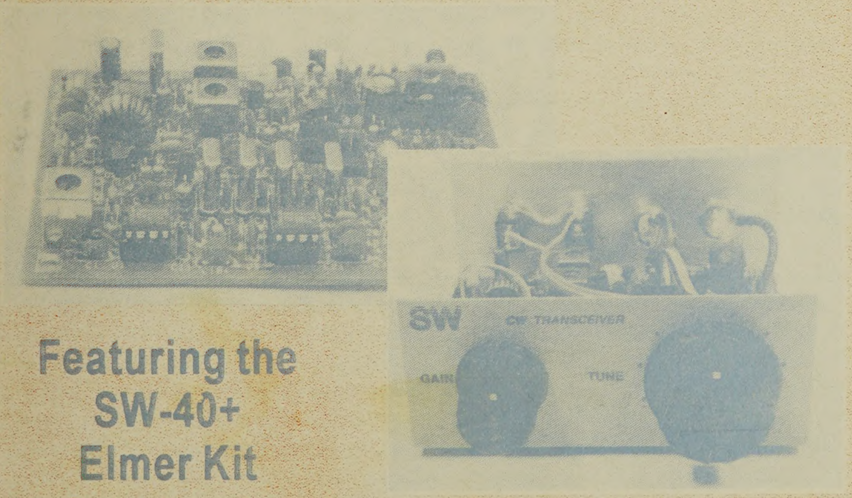
QRPp

ELMER 101 SPECIAL ISSUE

- Complete Lessons
- Circuit theory and analysis
- Test Bench procedures
- Step-by-step construction
- Build & Learn



Get the Workbench ready!



Featuring the SW-40+ Elmer Kit

Journal of the Northern California QRP Club

Table of Contents

From the Editor

by Doug Hendricks, KI6DS
862 Frank Ave.
Dos Palos, CA 93620
ki6ds@dpol.k12.ca.us 209-392-3522

You will notice something different about this issue. It is devoted entirely to one subject, the Elmer 101 Project. To my knowledge, this has never been done before by any of the QRP journals, or ham radio magazines. I did it because I know that there are a bunch of subscribers to QRPp who don't have internet access, and also to preserve on paper what is now only available from several different sources on the internet. The next issue of QRPp will go back to our regular format, and I hope that you will indulge me this one time.

And speaking of the internet, I have been remiss lately in not mentioning in QRPp the existance of my favorite web site, the NorCal Page, which is run by Jerry Parker, WA6OWR, who puts in hundreds of hours provinding up to date QRP information to NorCal members, and attracting new members to the club. The URL is:

http://www.fix.net/norcal.html

Check the page regularly for the latest up to date information on NorCal projects, reports on monthly meetings, and some great pictures. Jerry also provides links to a ton of QRP information from other sources.

I would like to end by saying thankyou to all who purchased the NorCal 20 kits. They sold out in 18 days. That must be some type of record. If everything goes right with parts aquisition, we should be able to ship the last week or so of October. But that depends on everything going just right. We will take the time necessary to do the rig right. Dave Fifield and all of the NorCal 20 team are burning the midnight oil to ensure that you get the best possible radio. Thanks again for your support. Hope to see all of you at Pacificon 98 for the QRP Forums and Fun. 72, Doug, KI6DS

Elmer 101: How it started

By Doug Hendricks, KI6DS
862 Frank Ave.
Dos Palos, CA 93620

Last January there was a discussion on the internet qrp reflector **qrp-l@lehigh.edu** about the need for a course that would enable the builders of kits to understand just exactly what was going on in the rig, to explain not only what part was being used where, but why.

QRP-L has a history of taking simple subjects and making them horribly complicated, and thus many good ideas die on the vine. I have seen this happen several times when it was suggested that the guys on QRP-L design a radio and kit it. Somthing that I don't believe will ever happen, because there are just as many different ideas as to what the rig should be as there are subscribers to the list. You just can't get 2500 qrpers to agree.

The more that I read, the more that I could see that this was a good idea, and that it should not be allowed to die the horrible deaths that other good ideas on the list had suffered. I thought about it, and decided to post to the net a message that contained the following points that needed to happen for the class to work.

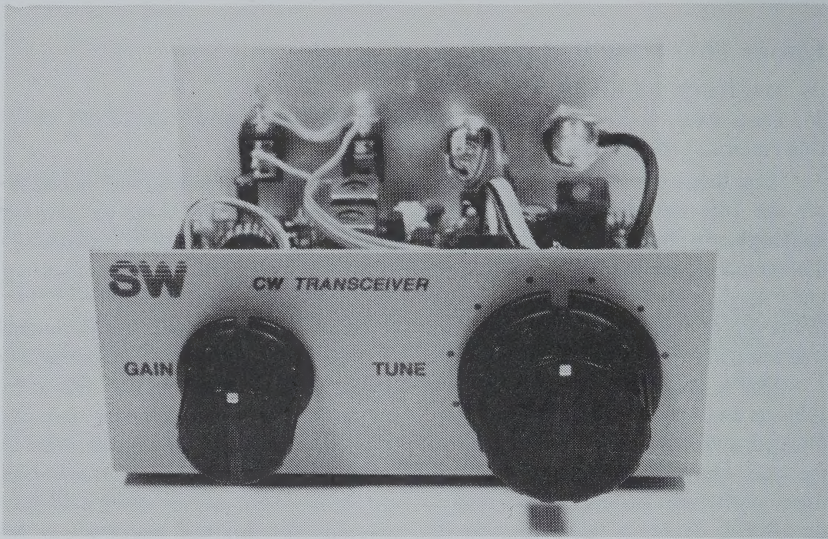
1. Contact Dave Benson and see if he can supply the group with kits for the 40-40 in quantity.

2. We need an instructor to be in charge. He doesn't have to teach the whole class, just be in charge. Qualifications: Knowledge of the subject, ability to communicate, organizational skills, and able to finish the job.

3. The class would consist of going through the schematic part by part,and explaining the purpose of every part. We would build the radio as we go, and the course would be on QRP-L, with someone putting each day's lesson on a web page, so that we will have it later for the newbies. As we go through the schematic, the instructor might have us build a simple oscillator and then try different resistors to see what happens when we change the bias, etc.

4. We have had numerous threads on this list that have died because of lack of action. Here is a plan. Who will step up to the plate? The most important question now is how many people are interested in doing this. I don't have time to take this on, but is there someone who is willing to take private email and keep track of the number on this list who are willing to order the kit from Dave and participate? If so, make yourself known. Post to the list that you will take a list of those who will buy a kit and participate, and be sure to give a deadline, say midnight Wednesday, so those on the digest have a chance to email you. Then report back to the list the number. Maybe there are 100, maybe 5 who knows? Lets get started. When we have a number, then we can ask Dave about delivery times, etc. 72, Doug, KI6DS

This was the extent of my involvement. I made a suggestion and Mike Maiorana stepped up to the plate and volunteered to organize and run the project. He did a wonderful job. Chuck Adams contribution of the idea to use the Small Wonder Lab's 40+ as the radio for the course was brilliant. Dave Benson even updated the design of the 40-40 to the SW40+ so that we could have a better kit. In fact, you can still order the kit, the SW40+ from Small Wonder Labs. See pages 4-5 for ordering information. Now, lets get started on building and understanding a CW transceiver. All of us hope you learn from our efforts.



SW+ Transceiver Series

The SW+ Series is Small Wonder Labs' latest product offering. This CW transceiver board kit is an updated version of the classic '40-40' transceiver first fielded by the New England QRP club. This design was subsequently published in the November '94 issue of QST and serves as a centerpiece in the ARRL's recent anthology, 'QRP Power".

Design features:

1. Single-board transceiver, 2.8 x 4.0" (7 x 10.1 cm)
2. Commercial-quality board, masked and silkscreened
3. True VFO coverage- 35-40 khz
4. Superheterodyne design, with crystal filtering

The SW+ series retains much of the original receiver design, while augmenting the transmitter design for improved stability and output power. The following design changes went into making a classic even better:

Added power supply reverse-polarity diode. The number of toroids to wind drops from 8 (in the orginal) to 5. This pays off in faster and simpler assembly. The diode bridge T-R switch is replaced by a series L-C type. This drops receiver current draw from 22 to 16 mA and improves image rejection.

Better yet, there are several more reasons to consider the new SW+ series:

Improved board- The printed-circuit board is now double-sided and solder-masked both sides. Component silkscreening is improved, and all parts are identified on the silkscreen by both outlines and reference designators.

Upgraded documentation- The manual is expanded to incorporate additional information for beginning builders. The manual now also breaks construction into groups to allow less-experienced builders to proceed in easy stages. The manual itself has undergone a 'facelift, with new figures added.

I've saved the best news for last- The price hasn't changed! I'm able to offer the SW+

at the pre-existing pricing, and plan to do so for some time. This is afforded by the price breaks I can get on larger parts quantities.

The pre-existing SW-40 and its siblings are now being retired. All future orders for this kit will be filled with the new series, and I'm confident that you'll be pleased with the differences.

Price and Schedule:

The SW+ series is currently available for 80M, 40M , 30M or 20M. The SW-40+ and SW-30+ are $55 (postpaid) to US and Canada. Overseas customers please add $5, this includes airmail shipping. (CT residents please add $3 state sales tax)

Enclosure Kit:

An enclosure kit is available for the SW+ series. This resembles the earlier SW-series enclosure with a number of improvements.

1. The kit includes a customized Ten-Tec TG-24 enclosure, which measures 4.1"(W) x 4.0"(D) x 2.0"(H). It's a handsome two-tone (black/grey) finish, and is now silkscreened on both front and rear panels.
2. The rear panel complement has been revised with respect to its predecessor. The phono ('RCA') has been replaced with a 3.5mm (1/8") jack- this facilitates use with Iambic paddles should you wish to install a KC-1 (Wilderness) or TiCK (Embedded Research) keyer kit. The pre-existing DC power plug is replaced with a 2.1/5.5mm coaxial connector; this connector is common to other QRP kit designs.
3. As with its predecessor, the SW+ board snaps into 4 mounting posts, so there's no hardware to work loose in the field. Both the new and old board versions may be installed in this enclosure.
4. Interconnect to the connectors and controls comprises a set of .100" locking headers and mating terminal housings. The enclosure kit provides preassembled harnesses to make wiring a snap- you simply cut the harnesses to length and solder to the connectors and controls. The documentation which accompanies the enclosure kit provides illustrations and clear directions for enclosure assembly and final hookup.

Price and Schedule:

The SW+ enclosure kit is $35 plus shipping ($3 US and Canada). Overseas shipping is $10 by airmail. Note: the complete overseas kit price is $100 including shipping.

Technical Support- Each manual contains telephone, US mail and e-mail address information. All requests for assistance are answered promptly. If I can't bring your Small Wonder Labs product to life properly by phone or otherwise, troubleshoot and alignment service is available for a reasonable flat fee. Missing or damaged parts are replaced on a prompt "no-questions asked" basis. When you're ready to order, send personal check or money order (US customers) to Small Wonder Labs at the address below. Canadian customers may use a US dollar account or postal money order. Overseas customers can send an international postal money order, cash (if concealed and sent via registered letter), or contact me for other means of payment. All prices include air mail shipment.

Dave Benson, NN1G
Small Wonder Labs
80 East Robbins Ave
Newington, CT 06111
bensondj@aol.com

The Elmer 101 Project: Building the SW 40+

by Mike Maiorana, Glen Leinweber, Chuck Adams, Dave Benson, Mike Gipe, Paul Harden, Bill Jones, Gary Surrency & a class of thousands of QRP-L readers.

First, READ THE MANUAL! Dave has done a great job on the manual and it contains tons of useful information. Don't worry if the "Theory of Operation" stuff goes over your head, that's why we are doing this.

Inventory your kit. If you are missing any parts, double check then contact Dave Benson for a replacement. Don't wait to do this step, it will save you grief in the long run. Also use this opportunity to familiarize yourself with identifying the components. If you have trouble identifying the components after reading the manual, post your question to qrp-l@lehgih.edu (with the Elmer101 subject header)

Get the soldering iron, solder, desoldering braid and sponge ready. Do you have a 12-14 volt power source? It needs to be able to supply at least 500mA to the circuit. You should avoid using switching power supplies (like out of a computer) as they are electrically noisy. A small 12 volt battery should be fine, 8 D cell batteries in series would also work. You can buy a wall transformer that will power the rig, but they are pricy.

There are some parts that you need to complete the rig that you can get from RadioShack listed here:

1) 5k pot RS#271-1714
2) 100k pot RS#271-1716
3) headphone jack RS#274-249
4) key jack RS#274-247
5) antenna jack RS#278-105 (If you want a BNC antenna antenna connection.)
6) power jack RS#274-1569
7) power connector RS#274-1582

The only one of these that you need ASAP is #2 as it will be used in the VFO section (next).

You will need some small hookup wire for testing and final assembly, 20 gauge stranded insulated should work well.

Check your power source output with your volt meter and verify that it is at the proper voltage. First we will build up the power supply on the radio. This is the simplest part of the whole project and a good place to start (no power, no RF).

Also, please, speak up if you need anything! Also feel free to comment on how we may run the class better (be careful, you may be volunteering). All questions are welcome.

Power Supply

This section will cover the DC power supply section of the radio. It has the job of supplying the radio with a useable D.C. power source for the different circuit modules. There are two basic voltage supplies on the rig. There is the unregulated D.C. supply and there is a regulated 8 volt supply.

First find the following components in your kit.

D13 - 1N4001- Diode
C112 - 220uF electrolytic capacitor
C102 - .01uF ceramic capacitor
U2 - 78L08 - 3 terminal voltage regulator

As stated in the manual, many components are polarized. In other words, it matters which direction you install them. Diodes, I.C.'s and some capacitors are examples of polarized components.

Install the above components. PLEASE double check polarity and position before soldering. Reference page 11 of the manual for details on the diode install.

Connect 2 pieces of hookup wire between the power supply connections on the

circuit board and your power source. The - supply goes to ground (J4 pin 1) and the + supply goes to the +12 volts (J4 pin 2). You need to be able to easily turn this on and off, so make sure these connections are easily removed. (Don't build with the power on!!!!)

Voltages are measured with reference to ground unless otherwise specified. This means that the black (-) lead of your volt meter should be connected to the board ground, along with the power source (-) connection. You can connect the meter black lead to the power supply (-) connection to make the following measurements.

J4 pin 2 should be the same as the power source + terminal.

Cathode of D13 should measure the Power source voltage (Vps) - D13 dropping voltage (about 0.7 volts). The diode dropping voltage will vary with device and with the amount of current passing through it.

Output of the U2 voltage regulator should be about 8 volts (between 7.7 and 8.3 volts). A convenient place to measure this is at pin 1 of J2.

If you get these readings, you have built the power supply section correctly. Now on to the circuit description.

As you all know, a diode allows electrical current to pass in one direction but not in the reverse direction. When a diode cathode is more negative than the anode, it will conduct current. In this circuit, D13 is in series with the power source as it feeds the rest of the circuit. Its function is to protect the board from damage if the power leads are connected backwards. If the power leads are reversed the diode will be reverse biased and no current will flow into the circuit, saving all those little components from certain death.

The disadvantage to using a series diode for polarity protection is that you lose voltage (and power) in your power supply. There are other methods of polarity protection that don't significantly affect the supply voltage or power, but are significantly more complex or use a fuse. For a simple circuit this is an excellent solution.

C102 and C112 provide decoupling on the power rail. They provide a low impedance path for any AC on the internal power system. This keeps the supply a clean D.C. voltage with a very small A.C. component.

U2 is a low power three terminal voltage regulator. It is there to provide sensitive circuits a constant voltage. The input voltage to the board is not regulated. It can be anywhere between 12 and 15 volts. Lets just say we are using a battery for the power source. All power sources have an internal resistance (usualy small). As more current is drawn from the supply, the voltage drop across the internal resistance increases, decreasing the output voltage. So when we key up the transmit section and the current draw from the power source jumps, the voltage provided drops. There are certain circuit components that require a very stable voltage source. The VFO is one example of this. Imagine what the radio would sound like if the VFO frequency changed depending on how long the key was held down. (We call that chirp). This is a bad thing. So we use a linear voltage regulator to isolate the sensitive circuits from variations in the supply voltage.

The 78L08 device can take anywhere from 10.5 volts to 23 volts at its input and provide an 8 volt output. It provides overcurrent protection, short circuit protection and thermal protection (shut down if it gets too hot).

Remember that these devices are not perfect. We have looked at them up till now as a "perfect" device. They can only supply 100mA maximum. They can only dissipate a total of 700mW assuming the ambient temperature is <25 degees C.

Power dissipated by the device is equal to the voltage dropped by the device (15-8= 7 volts) times the current delivered. Notice that in this design that if the maximum current was drawn out of the device (100mA) at the maximum supply voltage (15 volts), the maximum power dissipation on the device is 700mW. Coincidence? I think not!

Also, variations on the input do appear at the output, although greatly attenuated. The spec is 48dB at 120Hz (full wave rectified line ripple). If my math is correct, a one volt change in input voltage will cause a15.8 microvolt change in the output.

In addition, the device itself uses a certain amount of power. Bias current is about 4mA. At 15 volts that equals 60 mW.

If you are curious, you can view the data sheet if your browser has a PDF reader.
http://www-s.ti.com/sc/psheets/slvs010e/slvs010e.pdf

Next, we build and discuss the VFO. It will probably be done in several separate sections. Stay tuned.

Why these parts?

I've received several emails with questions on the power supply circuit. I don't know if I can answer them completly, but this should serve to start the discussion.

All the questions centered on a theme. WHY did the designer choose this particular component?

Lets also keep in mind that many components are chosen for reasons other than function. There are many parts that could be replaced with a number of parts and function properly. Often the choice comes to economy, availability and stock. Dave may have 10,000 1N4001 diodes in his stock. That would move him to use these components in his designs anywhere possible. Physical size is also a consideration, will it fit on the board? Component choice can also be "designers preference".

D13, why a 1N4001? I think I can answer this one. What characteristics do you need in this particular application? It must be able to carry all of the power supply current through it (up to .5 amps). It must have a reverse voltage rating greater than the supply voltage (15 volts). It does not have to be fast, as it is not a detector or a switching diode. Capacitance really does not matter as it is normally forward biased. It should be inexpensive and readily available.

So, the 1N4001 fits these characteristics perfectly. It handles up to 1 amp forward current, 100 volts reverse voltage, characterized as a rectifier. And they are really cheap.

You could replace this component with any rectifier diode that has a PIV of 30 volts or more(input voltage plus the charged Power supply capacitor) and a forward current rating of 1 amp (0.5 amps plus surge current) or more.

C112 and C102, why these types and values? I think I can answer part of this question. C112 is a 220 microfarad electrolytic. C102 is a .01 microfarad ceramic. C112 is for filtering low frequencies and C102 is for filtering high frequencies (RF). The electrolytic has a low impedance to lower frequencies (120Hz) but has a substantial impedance to high frequencies, due to it's construction. This is why C102 is there. There are lot's of different types of capacitors with widely varying characteristics. Does anyone have a summary of different capacitor types and their characteristics? That would be a good article.

U2 78L08 Why? I believe this one was already answered in the original post. It keeps a steady 8 volts on sensitive circuits even if the supply drops to10.4 volts.

Why this particular part? It is a standard three terminal regulator. They have

been around a while. Take a look at the date on the top of the data sheet (January, 1976). From what I understand there are two types, the L series in the TO92 (plastic transistor) case, and the standard series is in a TO-220 (like the final output transistor, Q6). The standard series can supply lots more current. It is also more expensive, larger and has a higher bias current. Since the circuit requirements are for <100mA , the 78L08 is the correct choice. You could "roll your own" regulator with discrete components. However, at $0.25 each, and such a small size, I don't think you can beat the 78Lxx series.

I hope this is a start in the right direction to answer the questions posed. Let's hear from those "in the know" on the details I missed.

More Whys.

Here are the answers to the capacitor selection questions. I got this info from Dave Benson, the designer.

"Why do we use a 220 mF instead of a 100mF to filter the low freq junk?" *The 220 microfarad cap was the largest value available in that size package.*

"Why do we use a .01 mF instead of a .1mF to filter the RF?" *Yes, they'd all work, the .01 is cheapest.- I buy caps in lots of a thousand-the difference between $.02 and .05 fills up my car's gas tank a couple times.*

I do remember someone asking why the bypass caps on U1 and U2 were different values, and I saw some good explanations. The real story is that they would have both been the cheaper .01s but one of the locations was physically too tight and I upgraded to the smaller-package .1 cap!

I hope this clears this all up. Thanks Dave for the feedback. So, it should be clear that often a component choice is not made only on it's function. Economy and size are also a large factor.

Power supply rejection ratio error. The following text is in error in the power supply section. "Also, variations on the input do appear at the output, although greatly attenuated. The spec is 48dB at 120Hz (full wave rectified line ripple). If my math is correct, a one volt change in input voltage will cause a15.8 microvolt change in the output."

Sorry .... the power supply rejection ratio is in units of power, so the power is indeed reduced by 48 db. But, this corresponds to a 4 mV change in voltage. This is one of those famous 'db voltage' vs. 'db power' problems.

10^(-48/20) = 0.004

Schematic error - C102

I recently posted that C102 was at U2's INPUT side. WRONG! I had looked at the schematic to determine where it was, not on the board itself....

On the schematic, C102(0.01uf) is shown at U2's(78L08) INPUT. But how is C102 connected on the board? Its at U2's OUTPUT side. So if you measure the DC voltage across C102, you'll see +8v. Not +12v. Everyone should scratch out C102 on their schematics, and draw in a new C102 at the top of the tuning pot (J2, pin 1). -Glen VE3DNL

Lets get the VFO built.

Some quick theory first. A schematic of the VFO only is shown in Fig. 1. It is also available at
http://reality.sgi.com/adams/vfo.jpg
(Thanks Chuck)

VFO stands for variable frequency oscillator. The VFO determines the frequency at which the radio operates. The vfo signal is mixed with the incoming RF to convert the desired receive frequency to the IF (Intermediate Freq.) of 4MHz. It also determines the transmit frequency by basically the opposite function.

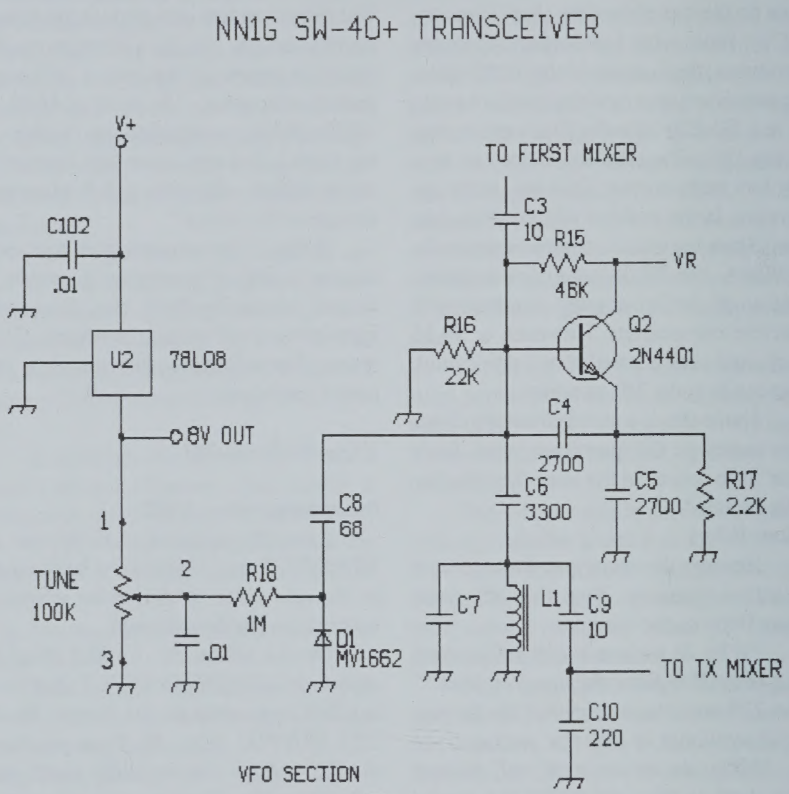


Fig. 1

There are 3 things that you need to make a VFO (or any oscillator) work. You need gain (amplification), feedback from output to input in phase, and a frequency determining device (or network).

Lets get it built. Find and install the following components (with the power off)

C2 47 pF
C3 10 pF
C4 2700 pF
C5 2700 pF
C6 3300 pF
C8 82 pF
C9 10 pF
C10 270 pF
C103 .01 uF
C105 .01 uF
D1 MV1662
D2 1N4148
L1 (You did finish it, right?)
Q2 2N4401
R15 47K
R16 22K
R17 2.2K
R18 1 Meg

Install a jumper between J2 pins 2 and 3. You can use a resistor lead that you trimmed from one of the above components. This temporarily takes the place of the tuning pot, which will be mounted later.

When you finish, double check the component placement. Also check your

soldering.

Fire it up! Put your RF probe on the base of Q2. You should see some RF voltage. Those with oscilloscopes or frequency counters should see an approximatly 3 Mhz signal here. The circuit will be calibrated later so don't install C7.

Get this portion built, and stay tuned for a circuit description, spice analysis and some experiments.

A VFO Experiment

Here's an experiment to try on your freshly-minted VFO. It'll show some quirks of oscillators, and how they differ from normal amplifiers in the way they're biased.

Set your meter to "DC volts", and measure the base voltage and emitter voltage of Q2. It gets kinda crowded at the transistor pins, so you might want to measure these voltages across R17(2.2K) and R16(22K) instead.

Here's what I get: Q2's base voltage (R16, 22K)........2.17V Q2's emitter voltage (R17, 2.2K)....2.4 V

Ok, what's strange about this picture? The BASE is lower than the EMITTER!! How the heck can Q2 work when its base to-emitter voltage is reverse biased? In a normally working amplifier, the base must always be about 0.6 volts HIGHER than the emitter!

Now you may get slightly different results, because your meter will "load down" the oscillator differently than mine. But you will still likely see the strange situation where Vb < Ve.

If you want to extend this experiment further, turn off the power and tack-solder a short jumper temporarily across the toroid (L1). This will keep the circuit from oscillating. Now power up and re-measure the DC base voltage and emitter voltage: Q2's DC base voltage.......2.13V Q2's DC emitter voltage....1.5V

OK, that makes sense. The base is now about 0.6 volts higher than the emitter. With 1.5V across the 2.2K emitter resistor, the transistor is biased at 0.68 mA. emitter current. Collector current is about the same. Now its biasing the way an amplifier should...but its not an oscillator anymore. (Unsolder the short jumper now, before you forget)

So what's going on with the oscillator - how can it work when the DC base voltage is less than the emitter? Well, the key here is that there are large AC waveforms (at 3MHz.) at the base and at the emitter too. The AC amplitude at the base (3.8v p-p) is larger than the amplitude at the emitter (1.9v p-p). The base voltage DOES climb 0.6v higher than the emitter, but only during a short part of the cycle: right at the positive peak. For the rest of the cycle, the transistor is biased off, because the emitter voltage is lower than the base. So the transistor conducts current in short pulses, maintaining the oscillations by jamming a current pulse from the emitter into the resonating L-C components every cycle.

You don't see these pulses, because of the "flywheel" effect of the 3 MHz. resonant LC components. All the AC voltages look like sinewaves as a result. All the transistor currents aren't sinewaves but are pulses.

Its also important to realize that AC voltages and DC voltages co-exist in any amplifier: interpreting what your meter is telling you can be tricky (especially when the AC volts are bigger than the DC volts).
Glen VE3DNL leinwebe@mcmaster.ca

SW40+ VFO

A 3 Mhz. oscillator supplies a sinewave to both receive mixer and transmit mixer. This note describes how Q2, a Colpitts oscillator is biased, how it begins to oscillate, and discusses some of the component selection. PSPICE is employed to illustrate circuit operation. The coupling

circuits into the two mixers are described as well.

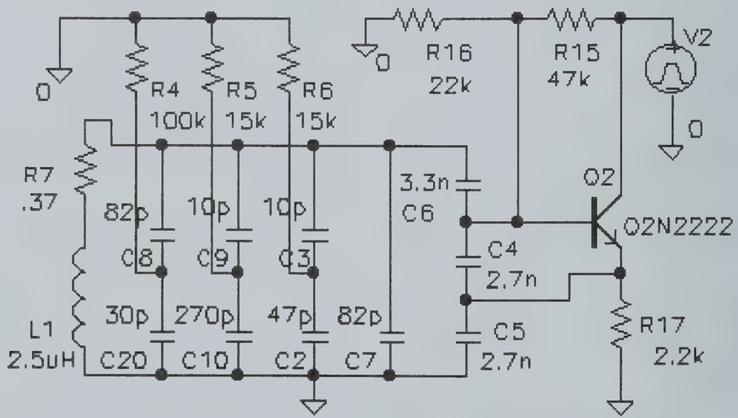
Biasing

Only three resistors are involved in the biasing of Q2: R15(47K), R16(22K), and R17(2.2K). Q2's base-bias point is set by the resistor voltage divider R15, R16. From the 7.4 volt collector supply, you'd expect the base voltage to be 7.4*(22)/(22+47) volts or 2.35v. Instead, Dave shows it to measure 3V (pg.19 of the manual) - why the difference? The answer is that Q2 is oscillating. It is actually operating in a very non-linear way. Should you prevent oscillations (by shorting out L1 with a short jumper), you'd find that the base voltage would settle much closer to the calculated 2.35 volts.

In a PSPICE simulation, Q2 was started initially with its collector supply at zero volts. It was then raised up quickly to 7.4 volts. The base and emitter voltages were plotted against time (with zero seconds corresponding to the moment that the collector supply was raised). The base voltage starts at zero volts because C4 (2700pF), C5 (2700pF) and C6 (3300pF), are all discharged to zero volts. These capacitors all start to charge through the base-bias resistors. About 30 microseconds later, Q2's base rises to +0.6v and Q2 now begins to conduct current from collector to emitter. The emitter current charges C5(2700pf) slightly faster, and C4 doesn't charge much more because the base-to-emitter voltage track together (a 0.6 volt differen

Somewhere between 35us. and 45us. Q2 starts to oscillate at 3Mhz. The amplitude starts very small from random noise. All the while, the capacitors continue to charge. The 3Mhz. amplitude builds quickly, because Q2 has a lot of gain and pumps up the resonant circuit a little more cycle- by-cycle.

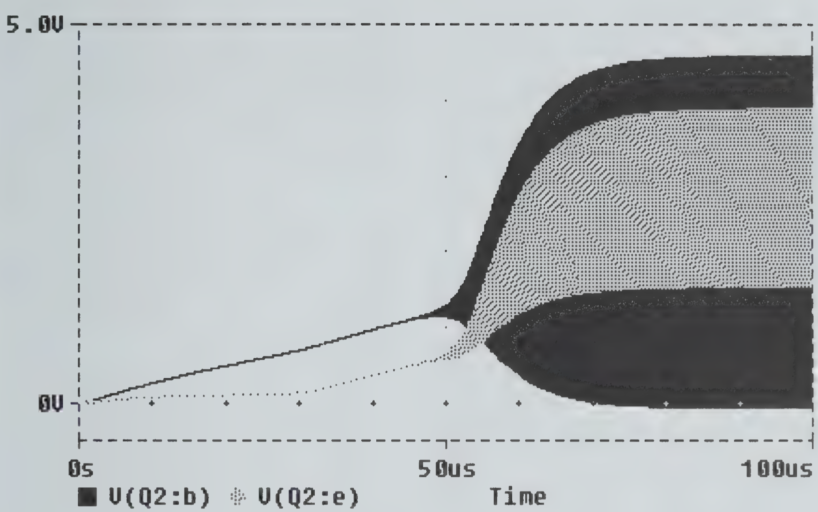
At some time around 50 us., the oscillating amplitude has grown so large that



SW - 40 VFO

Q2 begins to operate non-linearly. Up until this time, Q2 has been conducting current from collector to emitter all the time. Now however, on the bottom peaks of the oscillation cycle emitter current zeroes-out. For the fraction of the oscillatory cycle where emitter current is zero, Q2 is not contributing any current to build the amplitude higher.

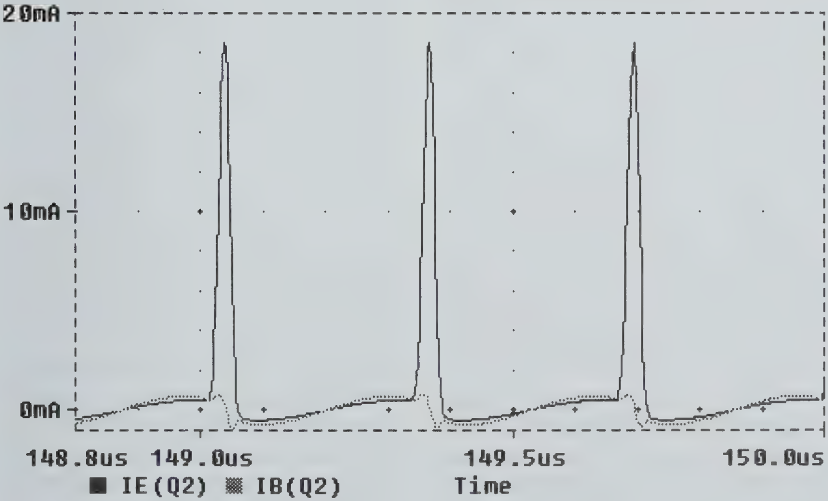
Between 50us. and 80us. Q2's emitter current shifts from a more-or- less sinusoidal



PSpice Simulation of VFO Envelope

waveshape to a pulse waveshape. The fraction of a cycle where Q2's emitter current is zero grows longer and longer. But Q2 still conducts enough (and has enough gain) to continue to add to the oscillation amplitude. So amplitude still builds, but at a slower rate.

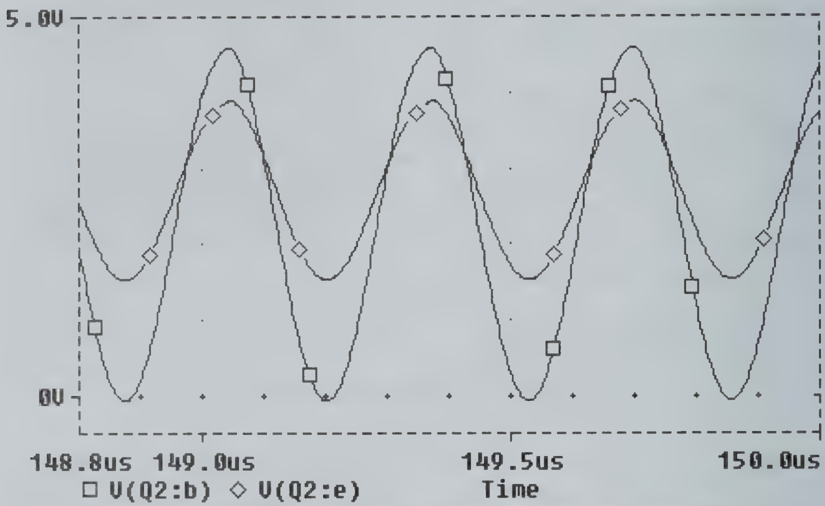
Now we come to a fine balancing act. Q2 is cut-off for most of the 3Mhz. sine-wave cycle. For the short time it is conducting, it contributes enough of an emitter-current "pulse" to maintain the oscillation amplitude. Here, the energy lost every cycle (dissipated in lossy components, and delivered



PSpice Graph of Current on the emitter and base of Q2

to the two mixers) is exactly replaced by the emitter current pulse.

After about 80 us., this balancing act is in equilibrium. According to the PSPICE simulation, the emitter conducts only 14% of the time, at the positive peak of the AC waveform (on the base). The rest of the time Q2 is off - very little current flows either into the base, or between collector and emitter. While the emitter current looks like a string of short, sharp "spikes", the base voltage and the resonant voltage at L1 are clean sinusoidal waveshapes.



PSpice Simulation of the Voltage on the emitter and base of Q2

Which components form the 3MHz. resonant circuit? The only inductance in the circuit is L1 - it is the inductive reactance of the resonant circuit. The associated capacitive reactance is distributed among capacitors C2 to C10, plus the capacitive reactance of the varactor, D1. Much of the resonant current flows through C4(2700pF), C5(2700pF) and C6(3300pF). The reactances of the other capacitors are relatively higher, and they contribute less to the resonant energy.

L1 has about 2.5uH inductance. To resonate at 3 Mhz. a parallel capacitance of 1126pF is required. To get to 3.04Mhz, a parallel capacitance of 1096pF is required. The varactor diode must supply the difference of 30 pF.

Dave "taps-off" some energy of the resonant circuit to feed 3MHz. signals into mixer U1 and mixer U5. He does so with a capacitive voltage divider. C3(10pF) and C2(47pF) feed about 1V p-p into the U1, the receive mixer. The RF voltage across C2 will have a sinusoidal waveshape. This voltage divider allows U1's internal biasing to set the DC voltage point of its oscillator input, without disturbing Q2's own DC bias setup.

U5 (the transmit mixer) requires a much smaller 3Mhz. signal, because it is injected into the high-gain port rather than the oscillator port (as is done for U1). The small reactance of C10(270pF) only allows 0.2V p-p into U5.

Mike Gipe, K1MG, talks about capacitors and varactor diodes.

In theory, all hams are slim, handsome, great conversationalists at any cocktail party, and have over-achieving kids. In practice, ...

In theory, all capacitors are pure and stable, strictly obeying the impedance formula, x = 1/2piFC. In practice, they change value with temperature, they have lossy dielectrics, they have resistive contacts, and they also have a bit of inductance in the leads and plates.

If you remember the impedance formulas from your ham exam, you will recall that the impedance of a fixed value of capacitance decreases as you increase the frequency of the signal you are passing through it. The impedance of a fixed value of inductance increases with frequency. What happens, as you increase the frequency of the signal, to the impedance of a typical ceramic capacitor which has a little bit of stray inductance? At low frequencies, the inductive part contributes very little impedance, so the capacitive impedance predominates — that is, it behaves like a capacitor! As you increase the frequency, the capacitive impedance decreases and the inductive impedance increases.

Would there be a particular frequency at which the capacitive impedance equals the inductive impedance? Absolutely! And if you increase the frequency still further, the inductive impedance becomes higher than the capacitive impedance, and your poor capacitor behaves just like an inductor! The frequency at which both impedances are equal is known as the self-resonant frequency. This frequency is set by the materials used and the construction of the capacitor. It can also be affected by how the capacitor is installed, which is why all the kit instructions tell you to mount the cap very close to the board, with the minimum lead length necessary. In general, for capacitors of the same type and general construction, the larger value cap will have a lower self-resonant frequency.

The ideal bypass component would have infinite resistance at DC, and zero impedance everywhere else. Unfortunately, this component is not yet available, but a capacitor comes close. It has very high impedance at DC and decreasing impedance at higher and higher AC frequencies — this is, until you hit its self-resonant frequency. At that point and beyond, it becomes less and less effective as a bypass component because its impedance is becoming larger. The capacitor is a very effective bypass at some frequencies but useless at frequencies somewhat above its self-resonant frequency.

How do we handle this problem? If we use a large value cap which is a good bypass at low frequencies but useless at high frequencies, and put a small value cap in parallel with it, the small cap will not be very effective at the low frequencies, but since its self-resonant frequency is much higher, it will be effective at the high frequencies where the big cap is useless. Simple!

If you pull out some data sheets for capacitors, you will often find graphs of this and other characteristics of the component (impedance vs. frequency showing the self-resonant point, equivalent series resistance, etc). Keeping all these practical limitations of real components in mind and adapting the design to accommodate them is part of the engineering process.

Incidentally, this is also the reason why you won't find a cell phone built with anything other than surface mount components. The self-resonant frequency of chip caps is much higher than leaded caps.

BTW - inductors have similar flaws. The latest QRPp has an interesting article about the RF choke used in the Back-to-the-Future project. It's an interesting education in the real impedance of an inductor at HF frequencies.

Smaller caps drift in proportionately smaller steps and are somewhat self compensating. I.e.., A 10% 10pF cap will drift up/down by 1pF. A 10% 100pF cap will

drift up/down by 10pF. The ultimate result of 10x10pF caps to equal 100pF may seem the same however, 5 of the 10pF caps may drift up and 5 of the 10pF caps may drift down resulting in a net change of zero whereas a single 100pF cap will always drift in one direction by the tolerance spec. Of course, it is improbable that you will get symmetrical results like 5 up and 5 down however, the odds are that you will NOT get 10, 10pF caps that all drift in the same direction. (You thought this was a science?)

I did not mention Silver Micas that are commonly used in VFO's. The Silver Mica film tends to be more rigid than ceramic film and are less likely to flex with the same intensity under the same proportionate degree of heat however, when they do flex, they flex at a higher ratio than ceramics. Plus, the drift direction is unpredictable. In tightly controlled temperature environments, they *can be* more stable than an NPO ceramic. In the real word this generally does not hold true.

On the topic of Varactors, they are virtually all 'good' BUT they are not the same as a variable capacitor. You cannot substitute them directly and achieve equivalent performance, but you can nearly always incorporate a varactor into a circuit and achieve good performance if you take all the effects into account.

These different effects include:

- temp coefficient
- junction conductance
- non-linearities with respect to instantaneous voltage, that is the sum of both the tuning voltage and the RF voltage at any time but especially the peak
- V.tune impeadance

Others have already addressed the different temperature coefficient, so I'll leave well enough alone.

All solid state juntions suffer from one significant difference from a 'real capacitor'. IF the RF voltage is large enough the diode junction will conduct during that portion of the cycle that exceeds the diode 'on' voltage. This introduction of non-linear effects includes changing the RF impeadance of the circuit, changing bias if the circuit is designed wrong, and changing the instantaneous capacitance and hence the tuning. This is to be avoided at all costs. sometimes you will see two varactor diodes in series with opposite polarity. the tune voltage will be feed to the junction of the cathodes. this is to eliminate conduction in the circuit. (one will always be non-conducting..) Preferably, the designer accounts for the peak voltages and ensures that the diode is not going to go into conduction.

In the same vein. A varactor diode (ALL!!!) exhibit non-linear relationships between V and C. This is still true with respect to the instantaneous voltage we talked about above. The C you really get is based on the sum of both the tuning voltage and the RF voltage at any instant of time. This results in spurious modulation, oscillators which will start and run for a few cycles, stop for some small portion of the RF cycle, and then restart (this is the voice of experience, and a REAL pain to find at Ghz freq's!!), and many other nasty behavior. Good designs will ensure that the instanteous variation does not substantatially deviate from the tuning point.

Although not a function of the Varactor, another potential problem introduced by the use of a varactor is the impeadance of the tuning voltage connection. this should be very high to ensure that no extraneous effects (circuit loading, stray rf/audio coupling, etc) will be introduced to the VFO. VFO's can make great mixers, product detectors, etc under the right conditions conditions. This line (V.tune) is generally a 100k resistor to each tuned

point for this reason. Mike Gipe, K1MG

Questions Generated by Mike's Article

Q: Mike, I am probably wrong but aren't D2 and C105 really part of the 2nd mixer circuit and not the VFO? Are they essential for the VFO to work or were they included in your list because you were following Dave's order of building?

A: D2 and C105 are part of the DC supply for the mixer chips and the VFO. Current flows from the DC power source through D2. It is filtered by C105 and supplied to the other parts from there. If you don't load these parts your VFO will get no DC power. As I understand it, D2 drops 8v to 7.4v. Also adds a little RF isolation for U1, U3.

Q: After stuffing the VFO parts, there seems to be an extra hole on my SW30+ pcb. Extra hole is connected to ground. (It is also on the SW-40+)

A: This is provided so you can solder a ground lead across the top of the three crystals in the IF filter (to be installed later). Grounding the filter cases prevents "blow by" which is strong external signals effecting the IF filter section.

Q: I would like to know the effect of using the 50K variable resistor in place of the 100K one that is used in the VFO section. As I recall the Radio Shack part number given for this was the number for the 50K pot. The schematic calls for a 100K pot. I have the 50K linear taper pot and the only 100K pot I could find at the local RS was an audio taper one.

Does using the 50K decrease, increase, or have no effect on the tuning range? And would one notice an appreciable difference in using the 100K pot that is an audio taper vs. one that has a linear taper?

A: A question came up about the tuning pot. The only function that the tuning pot, listed as 100K, has is to act as a voltage divider and provide the voltage to set the bias voltage on the MV1662 varactor diode. This in turn sets the VFO frequency. With 8V applied to the pot, 50K will give you 0.16mA and 100K will give you 0.08mA of current, so in the big scheme of things it doesn't make much difference. I used the 50K myself. BTW this is the current through the tuning pot only and just adds to the total current required for the rig.

Q: Now, many years ago, I learned all about Colpitts oscillators, but now I have forgotten.The function of C5 has me stumped. QRP Notebook says it is a feedback cap, but I don't see how that can be. Can either of you shed some light on this?

A: Let's first examine the 3MHz. resonant components. L1 is the resonant inductance, but resonant capacitance is due to a number of capacitances:

-C6 in series with C4 in series with C5
-C8 in series with varicap capacitance (d1)
-C7-C9 in series with C10
-C3 in series with C2.

These strings of capacitance all add up to resonate with L1 at 3MHz. A lot of the resonant current flows down the C6/C4/C5 string, since this string of capacitances represent the lowest reactance path of the bunch listed above. Now you could say that Q2 senses the resonant voltage at its base and injects pulses of current from its emitter into the resonant circuit in order to keep the oscillation amplitude up. So the junction of C4 and C5 is where Q2's emitter "pumps-up" the oscillations. That's the feedback point. Of course, Q2's emitter pulses are timed to re-inforce the sinewave shape that the resonant circuit naturally follows.

Q: Can somebody please help me understand what's going on by perhaps redrawing the circuit so all the series/parallel caps are more obvious, and then work through all the calculations that eventually lead to the value of the resonant circuit (near 3

MHz)?

A: OK, you've got the inductance right. Here's the string of capacitances redrawn. They're still kinda messy because there's so many. I'm assuming the varicap diode is 50pF. And I'm assuming that C7 (which is selected to get the VFO on the right frequency) is 68pF.

So let's do the math now. Start at the end, working out the series combination of C2 and C3:

10*47/(10+47) = 8.25pF, call this "Ca"

Do the same with C8 and D1:

82*50/(82+50) = 31.06pF, call this "Cb"

Do the same with C4 and C5:

2700*2700/(2700+2700)=1350pF

call this "Cc"

Now we'll add Ca, Cb, and Cc together since they're all in parallel:

8.25 + 31.06 + 1350 = 1389.31pF

This capacitance is in series with C6, so let's combine the two:

1389.31*3300/(1389.31+3300) = 977.70pF

call this "Cd"

Let's work out the series combination of C9 and C10:

10*270/(10+270) = 9.64pF

call this "Ce"

To finish off, we've got C7, Ce, and Cd all in parallel, so we add 'em:

68 + 9.64 + 977.7 = 1055.34pF

This is the capacitance that resonantes with L1, so we can work out the resonant frequency:

F = 1/(2*PI*SQRT(2.5e-6 * 1055.34e-12) = 3.0985 MHz. Pretty close.

3. How would I go about calculating the efficiency (Pout / Pin) for the oscillator circuit?

May I suggest that this isn't the kind of circuit where efficiency is important. Its not a power oscillator. The two loads (U1 mixer and U5 mixer) are both pretty high impedance and don't draw significant power from this VFO. And the DC power drawn by the VFO is a very small fraction of total transmitter power. According to Dave's DC voltage chart, emitter voltage is 2.5v DC. That's across the emitter resistor (R17) of 2.2K. So the VFO draws 1.14mA DC current from the supply. That's still a fraction of the 16mA receiver current. So you might think, "why can't we draw more power from the VFO, so that we don't need as much gain in the transmitter chain?". There's a good reason: U5 (the transmit mixer) can't handle big signals. It sets the limit on how much 7MHz. energy is available.

Q: More Math questions answered.

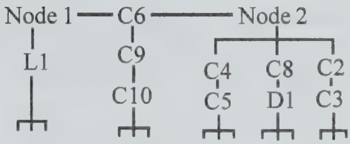
lets see if we can walk back through this.... first lets agree on a couple of conventions...

a) lets represent the series capacitance operation, 1/(1/c+1/c+...) as the operator (||), note that this is the same as resistors in parallel. Also two forms of exponential notation will help in ascii, 10 pf = 10E-12 =10*10^-12

b) since my ascii art is miserable to say the least, lets redraw the schematic in words the same way it would be done in a spice or ARD simulation, that is by series and parallel connections to numbered nodes. You'll probably want to do this w/ pencil and paper... To do this we'll agree that node 1 is the first node and we'll connect the inductor L1 there. A node is sim-

ply a point in the circuit where things are connected together. Node 0 is usually ground. The other numbers will fall out shortly. Since we're redrawing this to see the VFO LC pieces we'll ignore all others for just a minute....

L1 is connected from node 1 to node 0, or "IND 1 0 <value>; L1" C9 is in series with C10, connected from node 1 to 0 C6 is in series from node 1 to node 2 C4 is in series with C5, connected from node 2 to 0 C8 is in series with D1, connected from node 2 to 0 C2 is in series with C3, connected from node
2 to 0



we could also designate nodes between each series cap element (and would need to for any simulation input. Also, the note that the series combination of C9-C10, C4-C5, and C2-C3 are used to match impeadance by a technique called a capacitive split. A capacitive split works analagous to a tapped transformer. more on that later...

Now, the total capacitance can be written (remember the || notation) as:
Ct=(C9||C10)+C6||((C4||C5)+(C8||D1)+(C2||C3))
Using circuit values for the SW30+, and assuming
D1=120pF Ct(pf)=(9.64)+3300||(1650+60+8.25) = 1140 pf, or 1140E-12
F=1/(2*pi*f)(L*C)^1/2 and L= 1/ (2*pi*f)^2*C
So for 10.12 Mhz w/ IF = 7.68 Mhz, LO = 10.12 - 7.68 = 2.44 Mhz
L = 1/ ((2*3.14*2.44E6)^2*1140E-12) = 3.736E-6 or 3.736 uh

A VFO Quiz

1. Why is the diode capacitance a max with 0V at J2-2?

A. With no bias on the diode, there is a minimum thickness of the depletion layer between the P and N material. As reverse bias increases the depletion layer spreads, and like the plates of a capacitor, the farther away they are the lower the capacitance.

2. Why is the diode capacitance decreasing as voltage at J2-2 (pin 2 of J2) increases?

A. See above

3. Why is the diode reversed biased?

A. If it were forward biased it would have very little capacitance. Varicaps work in reverse bias mode.

4. What is C103 doing?

A. Preventing RF from the VFO from getting back into the 8 volt power supply.

5. Why is the voltage to J2 provided by the 78L08?

A. It is regulated to remove any variation from the power supply from changing the voltage thereby changing the vfo frequency. Ever heard a CW tone that chirped?

6. When rig is powered up, 8V is applied to the pot. Since we have R and we have I, then we have power applied to the 50K or 100K pot. It must thermally heat up and slightly change its resistance value. Why does this not affect the rest of the circuit, i.e. the voltage at J2-2 does not change over time?

A. Very little current leaves the pot at the center tap. A reverse biased diode in series with a 1 meg resistor draws very little current. That tells me that power is

dissapated evenly across the whole wiper surface of the pot. This way both sides change together with a net change in the voltage divider is zero

******DANGEROUS QUESTION ******
Do not try this at home.

7. Why must one be careful when using the clip lead and not connect it between pins 1 and 3 of J2?

A. This will short out the +8 supply to ground. In theory the regulator should shut down without being damaged, but I won't try it.

8. On some rigs, you might see a small trimmer cap in the Y5, RFC2, C28, and C29 part of the circuit. Why?

A. To tune out any deviation in the provided crystals. This circuit oscilates and mixes with the VFO to produce the transmitted carrier. It would fine tune the output frequency as it relates to VFO frequency.

PART 4: Keying Circuit and Transmit Mixer

Gather the following components.

U5 NE612 8 pin IC

PLEASE REFER TO DAVES INSTRUCTIONS FOR INSTALLING THIS PART. IT CAN BE INSTALLED BACKWARDS AND THIS WILL DAMAGE IT!!!

Socket for U5
Q3 2n3906 pnp transistor
D11 1n5236 7.5 volt zener diode
C28 47pf
C29 150pf (the round "ceramic" one, not the oblong one)
C108 .01mf
C109 .01mf
C110 3.3mf Electrolytic

ATTENTION, THE ABOVE PART IS POLARIZED AND CAN BE INSTALLED BACKWARDS IF YOU ARE NOT CAREFUL. ELECTROLYTIC CAPS HAVE A + AND A - SIDE. THE CAP HAS A STRIPE ON THE - SIDE.

C111 .01uF
R19 1K
R20 22K
R21 10K
RFC2 22 microhenry RF choke
Y5 4 Mhz crystal

Install all the above components after you double check the values. Install a small wire (3") with the end striped to J3-3. This is your test key. Double check your soldering and component placement. (Really, you should)

Ok, power it up. You should measure 0 volts DC on pin 8 of U5. Temporarily connect the wire from J3-3 to J3-1. This simulates a key down condition. You should measure about 7.5 volts DC on pin 8 of U5. Also, using your RF probe or oscilloscope, you should see RF on pin 4 or U5. Disconnect the wire from J3-1 and power down the rig.

Here is a quick theory of the circuit you just built. The transistor Q3 controls the power to the transmitter. When you ground J3-3 you turn on Q3 and allow power to flow to the transmitter circuit. In this case we only have the transmit mixer installed.

R19 and D11 form a voltage regulator for U5 which I think isolates it's power from the rest of the transmit chain. When power is applied to U5 it mixes it's own oscillator frequency (determined by Y5, RF32, C28 and C29) with the VFO signal on pin 2. The resulting output is the sum and difference products, plus lot's of other "unwanted" mixer products. The unwanted signals will be filtered out later. If you are looking at the output of U5 on a scope you will probably see a very messy signal, not a nice pretty sine wave.

What we are doing in this part of the circuit is deriving the transmit frequency

from the VFO by mixing it with the equivilant of the IF frequency. Remember that in this radio the VFO freq (3 Mhz) plus the IF frequency (4 Mhz) equals the receiver frequency (7 Mhz). The same thing is done to derrive the transmit frequency. The VFO (3 Mhz) plus the Y5 freq (4 Mhz) equals the transmit frequency. As our VFO frequency changes and our receive freq. changes we want our transmit to also change along with them. (Wouldn't be much of a radio if it didn't).

I'm sure this section will bring forth lots of questions on the venerable NE612 mixer chip. Glen Leinweber has provided a ton of good information on this little beauty.

If you would like to view the data sheet for this part it can be found here: http://www.us.semiconductors.philips.com/acrobat/datasheets/SA612A.pdf You need a pdf viewer to see it.

SE612 Integrated-Circuit Double-balanced Mixer

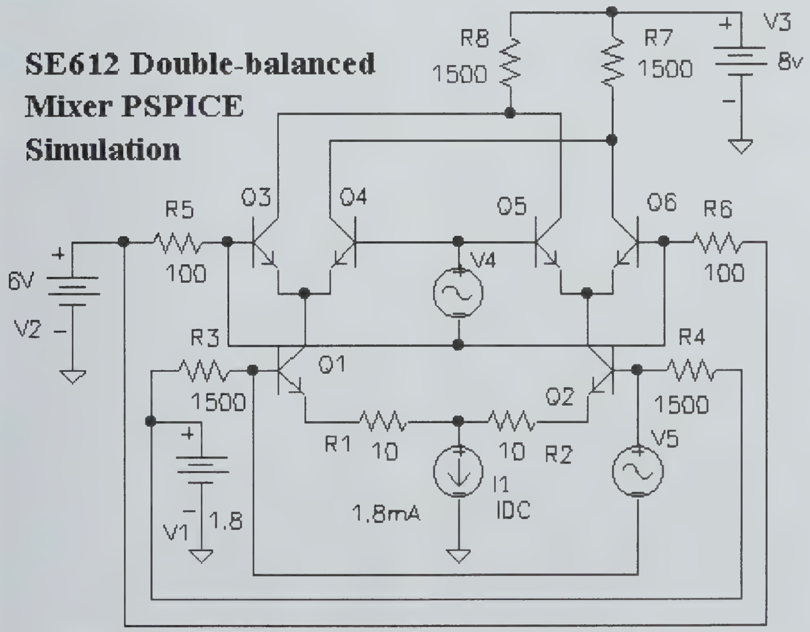
by Glen Leinweber VE3DNL

The SW-40+ radio employs three of these chips: one to heterodyne the incoming small-signal (7MHz.) RF down to the I.F. frequency of 4.0 Mhz., another to heterodyne the 4.0 MHz. I.F. down to audio, and another to heterodyne the VFO frequency of 3.0 MHz. up to 7 MHz. for the transmitting amplifier.

This note uses a PSPICE simulation to illustrate the mixing function of this chip. As an example, the transmitting mixer is examined to show how the 3.0 MHz. VFO is mixed with 4.0 Mhz. to get the desired 7.0 Mhz signal.

The term "heterodyne" refers to the mixing function, where two different frequencies are combined in a non-linear way

SE612 Double-balanced Mixer PSPICE Simulation



to generate an output waveform of a different frequency. Keep in mind that the process is non- linear: there is no way to linearly add two frequencies to get a third. An audio "mixer" is an entirely different animal: it simply **adds** signals together linearly. If it *did* output other frequencies, an audio mixer would be considered faulty, and in need of repair.

In the case of U5 (a SE612 chip), one sinewave input signal from the VFO at 3MHz. comes into pin 2, while the chip generates the other input signal at 4MHz. internally. The output waveform is available at pin 4 and/or pin 5. An internal Colpitts crystal oscillator generates the 4.0 Mhz sinewave (the crystal oscillator connections involve pin 6 and pin 7).

The heart of the mixer uses a circuit known as a *Gilbert Cell*. The output of this circuit is taken from R7 and/or R8. Inputs are represented as voltage sources V4 and V5:

A simple Gilbert Cell requires six identical transistors. Q1 and Q2 accept one of the mixer's input - in our case 3Mhz. from the VFO. The upper four transistors accept the other mixer input - in our case it is from the internal 4 MHz. Colpitts oscillator connected to pins 6.7.

V4 represents a 4 MHz. voltage source, simulating the Colpitts crystal oscillator connected thru SE612 pins 6, 7. V5 represents a 3 MHz. voltage source, simulating the 3MHz. input from the VFO. I1 is a DC current source, inside the SE612. If V5 were zero amplitude, half of I1's current would flow into Q1's emitter, and half into Q2. V5's input voltage unbalances current so that it flip-flops back and forth between the two transistors at a 3MHz. rate. Another way of looking at it is that Q1 and Q2's collector currents have equal amplitude, but opposite phase.

The real mixing action occurs at Q3, Q4, Q5, and Q6. The 4 MHz. signal drives the bases of these four transistors. Their collectors connect directly to SE612 output pins 4 and 5. Two internal 1500 ohm resistors provide a path to the +7.5v power supply. Notice how their collectors are cross-coupled together - part of the magic of double-balancing.

A simplifying assumption will show how mixing works: think of Q3, Q4, Q5 and Q6 as switches. In our case, Q3 and Q5 are "closed" when Q4 and Q6 are "open". Then the role reverses, with Q3 and Q5 "open" while Q4 and Q6 are "closed". The switching flip-flops back and forth at a rate determined by the 4 MHz. oscillator: each switch is "open" for 125ns., and "closed" for 125ns.

These switches direct the 3MHz. AC currents from Q1 and Q2 to the output pins, half the time to one, half the time to the other. The simple switching action described above is a VERY non-linear process, throwing chunks of the 3MHz. signals to one output or the other. The math is rather nasty, but in this process, other frequencies appear that are not simply harmonics of either the 3MHz. or 4MHz. input signals. We are entirely interested in one of these: at 7 Mhz. All others are unwanted.

A superb feature of the Gilbert-cell mixer is that neither 3 Mhz. nor 4 Mhz. signals appear at the output, provided that switching action is seamless, and the 3 Mhz. signals at Q1 and Q2 collectors are properly balanced. Ideally, only two frequency components will appear at the output: one at the difference frequency (4MHz - 3MHz = 1MHz) and one at the sum (4MHz + 3MHz = 7 MHz). That's the best we can hope for. Usually, these two dominate over a mess of other mixing products of lower amplitude.

The mixer still performs its task if Q3, Q4, Q5 and Q6 DON'T act as switches. Output will be smaller, but all we really

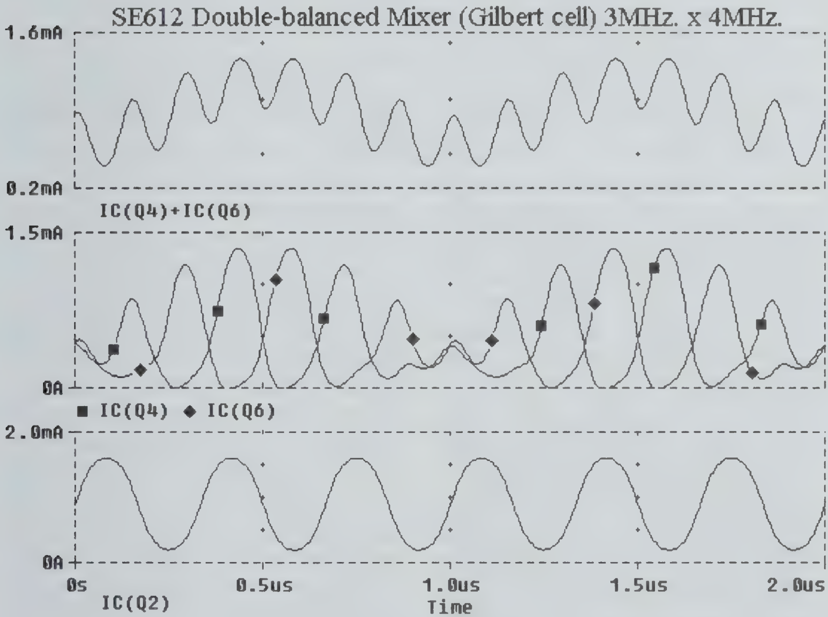
require of these four transistors is that they direct more current during their 125ns. “window” and less during the alternalte 125ns. “window”.

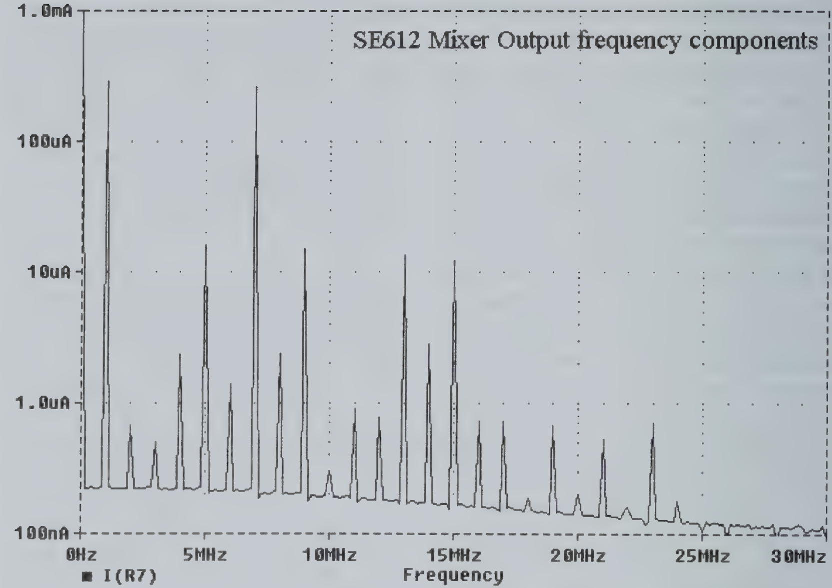
Now let’s look at the PSPICE simulation, showing two microseconds of the Gilbert-cell mixing action. Only currents of one branch of the cell are shown for simplicity. You can see that Q2’s collector current is almost a sinewave (the peaks are compressed a little) of 3MHz(bottom).

You can’t easily see much 3MHz. waveshapes at Q4 or Q6(middle), but the 4MHz. waveform is quite apparent, as is the longer 1MHz. wave. Note that Q4 and Q6 waveforms bottom out at zero current. These transistors are operating partly into the “switching” region. Even so, the combined collector waveform looks surprisingly good. Isn’t it marvelous how when the collector currents of Q4 and Q6 are combined, 3MHz. and 4 MHz. dissapear(top)? A really close, critical look at the output (top) will reveal some waveform compression. The output waveshape appears to be a combination of 1MHz. and 7 Mhz. sinewaves. The other output would appear similar, but out of phase. These output currents are translated directly into voltages by the 1500-ohm collector load resistors.

“Get OUT”, you say, “there’s nothing else there besides those two signals!”. Oh yeah? PSPICE can do a fourier transform of those outputs, showing the amplitudes of any and all frequency components that are there.

Yes, the 7MHz. and 1MHz. signals are the biggest, but there’s a murmuring mess of others too: many different combinations of 3MHz, 6MHz, 9MHz, 12MHz., and 4MHz, 8MHz, 12MHz, 16MHz. Of these we should be most concerned with





high-amplitude mixing products close to our desired 7 MHz. output, because these will be most difficult to knock down with a bandpass filter. For this example, the undesired frequeacies at 5MHz. and 9 MHz. will determine the extent of post-mixer filtering.

If we hit the mixer's input ports with smaller signals, these unwanted extras would die away to insignificance. But we'd have less output at 7 MHz. too. The SE612 mixer was really meant for low-signal receiver mixing, not high-level transmitter mixing. The result is puny output power, in need of amplification and/or filtering.

Don't expect the output waveshape from U5 to look as clean as this simulation. Balance is perfect here, not in real-life.

Part 4 Questions and Answers by Glen Leinweber

Q1. What is the significance of introducing the concepts of voltage source and current source in the analysis of this circuitry? I am more familiar with the concept of a voltage source...at least I think I know the "characteristics" of a voltage source.

A. Since Philips isn't very forthcoming about how the NE612 chip is internally biased, I've had to make assumptions about DC biasing of the chip. One assumption is that they use a DC current source to feed Q1 and Q2 (I1 in the schematic). Current-source biasing is very common inside integrated circuits since transistors can take less room and allow better and more stable performance than resistors. You could replace I1 with a resistor of about 660 ohms and get very similar results. This is a DC source. V4 and V5 are both AC voltage sources.

Q2. If V4 was based upon a 3MHZ voltage source, would the peak current through Q6 and minimal current flow through Q4 coincide with the timing of the peak current flow through Q2? I notice that the peak current flow through Q6 and minimal current flow through Q4 slowly change their

relative positions over time with regard to the peak current flow of Q2.

A. Yes, with both V4 and V5 at the same frequency, peak current would always be the same amplitude, and wouldn't shift with time. Let me expand on your suggestion: what happens when you mix 3 MHz with 3 MHz? You should get frequency components at sum & difference: at 0Hz and 6MHz. And indeed, if you make the one stipulation that V5 is 90 degrees out of phase with V4, you'll see a 6 MHz sinewave coming out the top. The peak/ minimum current flow thru Q4 & Q6 changes with time because the phase of 3MHz with-respect-to 4MHz changes with time. That's mixing at work.

Q3. "The mixer still performs its task if Q3, Q4, Q5 and Q6 don't act as switches. Output will be smaller, but all we really require of these four transistors is that they direct more current during their 125 ns "window" and less during the alternative 125 ns "window." (Quote from simulation) When I view the current traces for Q4 and Q6, is the minimum referred to in the quote from the analysis, when these traces "cross?" Likewise, the maximum referred to in the quote occurs when either Q4 or Q6 are at their respective peak values? I assume that it is no accident that the pattern of current flow for combined would reflect such a (summing strategy).

A. Perhaps its unfortunate that I've chosen signals levels part-way between "switching" action and the more linear "multiplying" action. Q4 and Q6 (Q3 and Q5 as well) "bottom out" at zero current on negative peaks. With V4 smaller, you'd see Q4 and Q6 display more symmetrical currents for +ve and -ve peaks - that's in the "multiplying" region. As shown, they're into the "switching" region partly.

Q4. The functioning of both Q4 and Q5 in their respective roles in the process is confusing when I consider the relationship of Q3/Q5 and Q4/Q6 to each of "their" respective 3MHZ transistors. Even though Q2 is permitting maximum current flow through Q6, do I assume that at the same instant Q1 (minimal current—180 degrees out of phase with Q2) is permitting "some" current flow through Q4? (although the analysis reveals very small amounts of current flow in Q4).

A. Ok, you're getting into the heart of mixing action with this question - you're close to seeing it all. But its hard to see the whole thing working at once. At the moment when Q6's current is maximum, you also see Q2 current peaks as well. So all of Q2's current goes up thru Q6 rather than Q5. Q5 is "off" Over on the other side (at this same moment) Q1 is suppling a meager current. Q3 is directing all of it up to R7, and Q4 is off. It'd likely help you to see the Q3/Q5 currents laid over those Q4/ Q6 currents. I had a look at this plot but it was too complex to follow, so threw out the Q3/Q5 currents. Keep in mind that I(q1) + I(q2) = 1.8ma DC (at every moment) Also I(q3) + I(q4) + I(q5) + I(q6) = 1.8 ma DC as well...at every moment. How these are distributed depends on the relative phase of the 3MHz and 4MHz signals. Its magical that you don't see any 3MHz or 4MHz stuff at either output - the magic of double balancing.

Q5. What is happening during the approximately .5 us, centered on 1.0 us with regard to current flow in Q4/Q6? It appears much different than the periods of time between 1.0 us.

A. There are moments (at 1us intervals) where everything is balanced. Naturally, because the phase of 3MHz and 4MHz sources is constantly changing these moments are fleeting. But at the moment of 0us, 1.0us, 2.0us...etc, Iq3, Iq4, Iq5 and Iq6 are all equal, 1/4 of 1.8ma. each. And Q1/Q2 are balanced too: Iq1=Iq2=.9ma.

Q6. Is the presence of 1 MHZ detected

through the slowly changing amplitude of the current flows in of Q4/Q6? Likewise the slowly changing amplitudes (?) of the combined IC?

A. Well, I'd say that the combined (Iq4 + Iq6) current shows the 1MHz presence very clearly. A little harder to see in the separate Iq4 or Iq6 waves, but its there.

Q7. The schematic reveals the bandpass filter is connected to each of the outputs of the mixer. Maybe this is to come, but what are the advantages of using both of the outputs from this mixer? Are these advantages part of the rationale for the use of a doubly balanced mixer?

A. Pin 4 and pin 5 of the NE612 have very similar wavshapes. You could use either one. But this chip suffers from having output signals that are rather small - they need a lot of power amplification to get up to 1.5 watts of 7 MHz out the antenna. Hey, its a "low-power mixer". So we should squeeze as much output juice as we can. Its easy to do: we simply take the output as a differential signal, amplifying the difference in voltage between pin 5 and pin 4. Its fortunate that the 7MHz signal is out-of-phase between these two pins. So we've doubled the output voltage - that's worthwhile. That's the only advantage - more signal.

May I suggest that it might help to view the four transistors q3, q4, q5, q6 as a bridge or ring. They're actually arranged in a similar fashion to the diode-ring balanced mixers. But they're rarely drawn that way.

The double-balancing trick is very similar in both circuits: With the diode ring, out-of-phase input signals are made with the multi-winding input transformer. In the gilbert-cell, its done with the Q1/Q2 differential amplifier. Then the mixing action is done by four devices in a ring: be they 4 diodes or 4 transistors. Perhaps the diode ring collects output signals in a more orderly manner (a single-ended output). Only two of these ring devices are "on" (dominant) at any moment. Great questions. Keep at it - its beautiful when you can see how the whole thing works.

I guess it's time for my 2 cents worth. I have worked for a leading manufacturer of mixers and have been exposed to their care and feeding.

1. Double balanced diode mixers.

There are many falvors of these depending on the need and your bank account. The difference in their performance is mosly a function of the diodes used and the local oscillator drive power necessary to have them function correctly. The most common variety uses +7 dbm (that is 5 milliwatts) of local oscillator (LO) drive. The Miniciruits SBL-1 is such a mixer.

The next LO drive level is usually +17 dbm (50 Milliwatts) or +10 dbm (10 milliwatts).....these guys cost about 3 or 4 times the +7 units.....Next comes the +23 to +27 dbm (200 to 500 milliwatt....yes that's a 1/2 watt) of local oscillator drive. These guys cost in the ball park from about 50 to 100 bucks each in small quantities. The difference in these mixers is their ability to handle signals without generating spurious responses. A company called Watkins Johnson makes many mixers and if you contact them they can provide excellent articles on mixer selection,

For average ham use say in a QRP rig's receiver, the SBL-1 mixer will provide very good performance.....If you spend about 10 or12 bucks more the SBL-1H using a +17 dbm (50 milliwatt) local oscillator drive mixer really work fine.

Time for a reality check....these mixers must be used properly for you to get the stated performance.....that is...all 3 ports (RF, LO, and IF) of the mixer want to see 50 ohm terminations.....The IF port especially is the most important. The LO port is easy....if you have some extra drive

available padded it with a resistive pad and then into the LO port. The RF port can be driven by a diplrxer which passes the frequencies you want and terminates all others in 50 ohms. The IF port can be terminated in a 50 ohm input impedance amplifier or a diplexer like the RF port.....I prefer a diplexer most of the time because it terminates the IF port in 50 ohms at all frequencies and dumpss the image freqs into the 50 ohm load.

I realize that many QRP rigs are offen powered from batteries and that receiver drain is a concern.....but if conditions are tough its nice to have a good mixer in the receiver front end.

Q. Can anyone please explain the details of the oscillator circuit connected to Pins 6/7 of mixer U5?

A. There's a transistor inside U5: base comes out at pin 6 and emitter comes out at pin 7. All the biasing is inside the chip. So you've got a Colpitts oscillator, similar to the VFO (Q2). In case you don't see it, here are the analogous parts:

C5 is analogous to C29

C4 is analogous to C28

These are the two critical parts to this Colpitts configuration, providing the positive-feedback path required by any oscillator. There's no single part that is analogous to the crystal. It is the resonating element, containing both inductive equivalent and capacitive equivalent of a resonant circuit. But what may be confusing is that 22uH choke. Let's take a closer look at its purpose. The short explanation is that its there to pull the crystal to a lower resonating frequency.

A little background: we need about an 800Hz offset between RX and TX. That's the "sweet spot" CW note that most folks like to hear. This offset is done by having U3's 4MHz. BFO crystal run at a frequency that is 800 Hz. higher than U5's 4MHz. crystal. The 22uH choke pulls Y5's resonating frequency lower by adding some inductive reactance to the crystal's own. Now 22uH is a lotta inductance. If it only pulls the crystal by 800 Hz. then Y5's own internal inductance must be absolutely HUGE. And it is. Inductive reactance inside the crystal is a fraction of a HENRY. That's astronomically big compared to Q2's resonating inductance of 2.5 MICROhenry. So you can see that pulling a crystal's frequency is really difficult. Now C28 and C29 also affect U5's oscillating frequency. You'll notice that C29(150pf) is much larger than U3's equivalent: C18(47pf). This was done to keep U5's oscillating amplitude low, in an effort to reduce U5's spurious frequency output at pins 4, 5. Having C29 large also helps lower the resonating frequency of Y5. But not enough. Adding 22uH to the crystal pulls Y5 down some more.

If you've got a working rig, with working RX and TX, you can try this experiment to explore the resonating frequency of U5's 4MHz. oscillator. Since some may be following the construction sequence on QRP-L (and not have U3 wired yet) I'll come back to this experiment later on.... Short out the 22uH choke (RFC2). This will raise the oscillating frequency of U5, much closer to the oscillating frequency of U3. You should either hear no sidetone, or a very low-frequency sidetone. This experiment demonstrates that the rig's TX/RX offset is set by the frequency difference between U3 and U5 oscillators.

One small detail on 602 type mixers... The biasing internal to the chip is sufficient for most applications. Occasionally you may find a resistor from the output (emitter) pin to ground. This is done to increase the bias current by lowering the effective emitter resistance by parralleling the internal resistance with the external

resistor. (R.effective = R.internal || R.external) A second effect is to extend the oscillator range or increase oscillator output, so you may also see this technique used when someone is using the mixer chip close to the specified freq. limits or in a tx mixer application. Glen VE3DNL leinwebe@mcmaster.ca

Part 5

As promised here is the next section of the Elmer 101 project. This is a short one and fairly simple so we should get through it quickly. Anyway, where we left off we had a hodgepodge of frequencies coming out of the NE612 mixer chip. Only one of the frequencies is the one we want to transmit, so we need to filter it out and stop the other unwanted signals from reaching our final amplifier. Dave uses a bandpass filter implemented with 10.7 Mhz IF transformers and some caps.

Gather the following components.

C30 47pF
C31 220pF
C32 47pF
C33 .01uF
T2 IF transformer
T3 IF transformer

Install these components in the appropriate places on the circuit board. Be sure to solder the "tabs" of the transformers to the board as they act as a shield and need to be grounded. Double check your component values and parts placement (Really, you should do this

You will need your RF probe, or an oscilloscope for the adjustment of this section. Connect your RF probe to pad of the base of Q4 (not installed yet). Turn on the power to the board and connect your temporary wire key between J3-1 and J3-3. You should see a reading on your RF probe. First adjust T3 for maximum reading on the RF probe. After you peak T3, adjust T2 for maximum reading. Go back and forth a few times to get the highest reading possible.

If you have an oscilloscope you should see a nice clean sine wave at about 7 MHz. Mine was 3 volts peak to peak at 7.111.42 MHz.

Here is what is happening in the circuit. The outputs of the NE612 (pins 4 and 5) are the differential outputs of the mixer. Remember the mixer discussion? You have the VFO signal at about 3 MHz and the internal BFO signal at 4 MHz being mixed at U5. The two main frequency components on the output are 7 MHz (desired signal) and 1 MHz (difference mixer component). Look at the mixer output frequency components at http://www.qsl.net/kf4trd/ne602.html . Notice that there are frequency components from the mixer as high as 25 MHz. We have to get rid of all of them except the 7 MHz one.

T2 and T3 are configured as a band pass filter, tuned with C30 and C32 for 7 MHz. The fine tuning is done by turning the slugs in the transformers. These IF transformers are designed to resonate at 10.7 MHz and contain internal capacitors, but the resonant frequency is lowered to 7 MHz by paralleling C30 and C32.

Also, when you finish up and adjust T2 and T3, you can prepare the primary of T4 as per the instructions on page 13. Don't install it in the board yet and don't install the secondary winding. We will need this for the next step.

Q: I'd like to hear more about Step 5. What are the advantages of the balanced vs. single-ended output configurations for the NE612? Whichever sort of output is used, is there an optimal termination for the mixer?

A: Using both outputs has the advantage of supplying double the signal for the next stage. The output impedance of the mixer is 3k ohms (from the data sheet).

Q: This is an exchange regarding the bandpass filter after the transmit mixer.

The original questions begin with

I'll start my questions on the Tx bandpass filter by quoting a section from the introduction in the rig's manual:

"Changed the Tx bandpass filter to use IF transformers. This configuration makes use of the differential outputs of the NE612. Filter bandwidth is increased considerably over the original configuration."

Is using the mixer in the differential mode any better? Why?

A. The outputs of pins 4 and 5 of the mixer are out of phase with respect to one.another. By using them both, as opposed to a single-ended configuration, there's twice as much signal to work with.

Q. I understand that in this filter we want a relatively broad bandwidth so any signal in the 40m band passes through unattenuated. I checked out the schematic for the original version (as seen in **QRP Power**), and he at first used an LC filter design. What was wrong with its bandwidth that it needed improved?

A. The original design was a little too sharp- it couldn't be modified much beyond the original bandspread with falloff becoming noticeable. I've simply broadened it a bit to make that adjustment a little less critical. Yes, a resistor could have been added to swamp the response. On the new version, the TX mixer source impedances (1500 ohms nominal at pins 4 and 5) as well as the emitter follower stage, establish loading on that filter. The value of coupling capacitor C31 also affects the filter response.

Q. I checked the Mouser catalog and see that one of these IF transformers sell for $0.65 in hundreds. So Dave probably saved a lot of real estate and maybe even cut the rig's cost by going to the transformers. But why are two transformers necessary; wouldn't one be sufficient? Don't two stages in the filter sharpen its frequency response, whereas the original idea is to broaden it?

Here were the key design considerations:

- Real estate
- Cost
- Easier asembly

It's possible to use a single tuned circuit and still meet the FCC requirements for spectral purity. Doing so, though, requires some pretty high impedances, which should avoided as a general rule with transmitter design- there's an increased susceptibility to unwanted feedback due to ground-loop problems. I didn't have the luxury of unlimited ground plane, although I was cognizant of ground paths, and elected for 'safe' over 'sorry'. Keeping the impedances reasonable and still retaining a good bandpass characteristic (good rejection away from the passband) calls for more parts.

Q. Also, very puzzling to me: the IF transformers are resonant at 10.7 MHz. But without knowing the values of the internal coil and cap, what's the process for determining that a 47 pF cap in parallel brings resonance down to 7 MHz?

A. The internal characteristics are easily inferred with a signal generator and scope. Once the resonance peak is found, extra capacitance is added in parallel and the new peak is located.

The resonance formula is

F(mhz)**2= 25,330 / LC

where L is in microhenries and C is in pF.

Doubling the capacitance (adding a parallel external capacitance replicating the internal one) moves the resonance to .707 of the original frequency. For these transformers, it's approximately 5 uH and 50 pF.

Q: Does the circuit following the mixer provide something like a 3K termination for the mixer? If so, how can we see this?

A. First, C33(0.01uF) has a very low impedance, so that T3 is coupled very tightly

to Q4 base. Whatever Q4's input Z is, that's what T3's going to see. Ditto with C34(0.01uF). Next, take a look at Q4's biasing resistors, R23(22K) and R22(10K). Their parallel combination is 6800 ohms. So T3 isn't going to see a load any higher than this.

Now comes the trickier part: determining the actual base input impedance of Q4. Whatever it is, it'll be in parallel with the bias resistors above, lowering the 6.8K load Z we have so far. Q4 is an "emitter follower". The impedance that we'll see (looking into its base terminal) is hie + hfe * Re where Re is the impedance from emitter to ground and hie is the base-to-emitter impedance of the transistor and hfe is the transistor's current gain (at 7 MHz.).

I'm going to make a guess at hie of about 300 - 1000 ohms. And I'm going to guess that hfe is about 25. If you look up the 2N4401 data sheet, you'll think I'm way off. They list hie=700 ohms (at 6ma. Ic). But they list hfe=150

The difference arises because these small-signal h-parameters were measured at 1KHz, not at 7MHz. At high frequencies, current gain drops off. At about 200MHz, it has no current gain at all.

What is Re? Maximum value is 500ohms due to R24. But in parallel with R24 is Q5 and its bias resistors. Bias resistors R25(2.2K) and R26(470) work out to 387 ohms. I'd estimate that Q5 has about the same Z as its bias resistors. So in parallel with R24 is 387 ohms, in parallel with another 387 ohms. So total Re is 500||387||387 = 140 ohms

So in summary, for Q4 we have:
hie = approx 700 ohms
hfe = 25
Re = 140 ohms
And its input Z is 700 + 25 * 140 = 4200 ohms.

This is in parallel with the bias resistors of 6.8K So the total Z that T3 sees is 6800||4200 = 2600 ohms

Now this is a minimum estimate, because I've assumed that the wiper of R24 is right up at the top. If its down lower, the total Z figure above will be higher, because Re will be larger. Dave has chosen bias resistors on the low side. This provides temperature stability, and also means that the load that T3 sees doesn't change wildly as you turn the R24 trimpot.

Q: Summary of transmit mixer problem.

I am building the SW30+ and have run into a problem. When I completed Lesson 4, I observed that Q3 was switching properly and that there was a complex waveform on pins 4 and 5 of U5. Feeling confident that all was well, I moved on to Lesson 5 and installed T2 and T3 and related components. The complex waveform observed on pins 4 and 5 of U5 are also present on the input of T2 but the output of T2 is flat. After checking all component values, orientation, solder joints etc. I finally concluded that T2 was probably working as designed, ie the input signal was outside of the bandpass of T2.

With that in mind I started backtracking and taking a closer look at the waveforms on U5 and made the following observations.

1) LO as measured at the base of Q2 is about 2.439 Mhz and 2.6V p-p.
2) LO as measured at pin 2 of U5 is about 0.16 V p-p.
3) Pin 7 of U5 is a beautiful modulated sinewave (NOT WHAT I EXPECTED). P-P voltage is about .05 V and it appears to be a signal of about 24-25 Mhz moduled by at about a 2.4-2.5 mhz rate. I was expecting to see a 7.68 mhz sinewave here. I have rechecked Y5, RFC2, C28, and C29. All values are proper, connections look good, etc.

What should I be seeing on Pin 7 of

U5? Assuming that what I am observing is not right, any ideas of what might be wrong?

TURNED OUT THAT THE CRYSTAL (Y5) WAS BAD. UP AND RUNNING NOW.......

Q: When I tweak T3 and T2 (while observing waveform on scope) I find the setting for T3 that gives me the largest amplitude to be with the slug turned all the way CCW (to the stop). T2's peak is roughly mid-way between the stops. T3 never really shows a peak - just gets larger as I go CCW. Is this okay?

Glen, VE3DNL, suggested that perhaps my scope probe was adding enough capacitance to the circuit to cause this problem. Following his suggestion, I went ahead and installed the Q4 buffer (and related resistors) and redid the alignment with my scope probe at the emitter of Q4 rather than the base. This effectively isolated the probe from the bandpass filters. The results proved to be quite interesting. There is now a definate peak about mid-range in T3. What is kind of weird is that as soon as I go slightly past the peak (CCW) the signal degenerates into a real mess. Dave Benson suggested a different solution which was to add 10-22pf capacitance across T3 (ie C32).

More on this problem

The procedural problem (measuring the waveform at Q4's base rather than at the emitter) did not result from any ambiquity in your manual. The Elmer 101 Part 5 had us stop prior to installing Q4 or any of its biasing resistors and then instructed us to measure the waveform specifically at the base of Q4 (no signal at emitter until buffer components are added in subsequent lesson).

I went ahead and put the buffer section in and while that helped it did not totally solve the problem. I was then seeing a peak more toward the middle of T3's range but when I hit the peak (going CCW) rather than seeing a reduction in amplitude what I saw was a degeneration of the waveform which required I unkey and rekey the transmit section.

I took your advice and put 22pf in C32 and found that threw T3 all the way to the CW stop. Reduced it to 10pf and found a nice solid peak near mid-range. That, however then caused T2 to hit its stop. I then put 10pf in for C30 and am pleased to say that I now have both transformers showing nice solid peaks near mid-range.

Even more....

Dave Benson suggested a different solution which was to add 10-22pF capacitance across T3 (ie C32). I will try that this evening and report the results. I'm betting that the two solutions togther solve the problem.

I assumed that you were looking at Q4's emitter, thus isolating the probe capacitance from the tuned circuit. As I look back, though, I didn't expressly say that anywhere in the manual- an oversight on my part. Scope probe capacitance will indeed upset the operation of that bandpass filter, and what you reported makes sense in light of the presence of probe capacitance. I'd recommend moving the measurement point to Q4's emitter and repeaking T2/T3- the extra capacitance I suggested earlier shouldn't be necessary.

73, Dave- NN1G

Part 6

We will build this up in two stages. First the RF buffer, then the driver. For the first section gather and install the following components.

R22 10K
R23 22K
R24 500 ohm trim pot
C34 .01uF ceramic (103)
Q4 2N4401

Follow all the standard installation

steps and check your work before proceeding!

Turn on the rig and connect your temporary key. You should see a nice RF sine wave at the base of Q5 (not installed yet). Remember that adjusting R24 will change the voltage from 0 volts to maximum. Make sure it is adjusted up. If you measured the output of the bandpass filter this reading may be a little lower. That's ok.

Just a side note. Any time you make a measurement of an electronic circuit you will effect it in some way, guaranteed. We try to minimize the effect by using the correct measuring device. Case and point, there were a few folks who had trouble adjusting T2 and T3 for a peak in the rf output. The reason for this was that the 0-scope probe was changing the characteristics of the circuit and changing resonance due to added capacitance and resistance. So, as we move forward keep in mind that the bandpass circuit is easily effected by circuit changes.

So we have this nice pretty 7MHz sine wave at the output of T3. The problem is that it is sensitive to circuit changes. So we must isolate it from the following sections. This is the job of Q4. It is configured as a common collector amplifier with voltage divider bias. The bias on the base of Q4 is set in the linear region of the transistor by R22 and R23. With a 12 volt supply this makes the DC bias on the base of Q4 3.75 volts. $Vr2=(12 * r2)/(r1 + r2)$

For information and an excellent explanation of bipolar transistor biasing please refer to chapter 8 page 20 of the 1998 ARRL handbook. It is a must read section!

In brief, the common collector amplifier (a.k.a. emitter follower) has some interesting characteristics. It has a gain of less than one (Hmmmm), a very high input impedance and a very low output impedance. So it does not provide any voltage gain, but it does isolate the bandpass filter and the RF driver section (next section).

If you tweeked T2 and T3 in the last section, put your probe on the hole for the base of Q5 (not installed yet). Now key up and see if you can tweek up the voltage with T2 and T3. I went through it again and was quite a bit off. This is because my scope probe changed the resonance of T3 and bumped the whole circuit a bit. My scope probe is rated at 10 meg ohms and 11.8 picofarads of capacitance.

Now while trying to peak T2 and T3 again I accidently bumped the ground shield on my scope probe to the top part of R20 (the side not connected to V+). Can you guess which component I fried when I keyed up, and why?

Ok, now on to part 2. First wind the primary of T4. Please follow the directions on page 13 of the manual. My experience was that 3" of wire was not enough for my coil. I would use 3.5" (I had to rewind mine, yuck). Install it and the secondary as per the instructions.

Gather the following components and install them:
R25 2.2K
R26 470
R27 10
R28 51
R29 51 (the books parts layout drawing has this incorrectly labled as 100)
C114 .1uF (104)
C35 .01uF (103)
D6 1N4148
Q5 2N4401

Check your work before continuing, paying special attention to T4. Put your scope probe on the base pin of Q6 (not installed yet). Key up and you should see your 7Mhz wave. Mine was distorted severly until I turned R24 down a little (I had it cranked all the way up). If you have an O-scope you will notice that the bot-

tom half of the wave is distorted. D6 clamps the negative half of the waveform at -0.7 volts. This part of the circuit has several things I don't understand but I'll try to explain what is obvious. The purpose of Q5 is to drive the final amplifier Q6 (not installed yet).

The final amplifier Q6 is a class C amplifier, which means the active device (the transistor) conducts for less than 180 degrees of the signal. The signal is kept going by the resonant circuit in the collector. Q6 just gives it a "kick" every cycle to keep it going. More on this when we put in the final amp.

R25 and R26 set the DC voltage on the base of Q5 at about 2.1 volts. This sets the DC emitter voltage at about 1.4 volts (2.1 - 0.7 base emitter diode drop). This sets the DC emitter current at 23 milliamps (1.4 / (51+10)). C114 bypasses R28 at RF frequencies so the effective emitter resistance at AC is 10 ohms which changes the AC operating characteristics (and increases AC gain??). Again refer to the ARRL handbook for details.

The load for this amplifier is the primary of T4 which is in the collector circuit. The manual describes it as an 8:1 transformer. I'm guessing that this ratio provides more current drive to the base of the final Q6.

Q: So what is accomplished by clamping the negative portion of the drive signal? It would seem that we're wasting valuable signal by shunting it to ground!

A. It's not really shunted to ground. The combination of C35 and D6 is a 'clamp' circuit. When the voltage at the base of Q6 goes very far negative, D6 begins conducting, and as such, it's charging up the .01 coupling capacitor. This negative side of the voltage swing normally contributes nothing to driving the class C stage, so there's no signal loss per se. On the positive side of the input signal excursion, the presence of this stored charge on C35 actually drives the base of Q6 a bit harder. The improvement is about 2 dB in terms of making the final easier to drive. You can verify this by setting the output power around 1.5 watts and removing the diode- you'll see output power drop.

The signal at the base of Q6 (without the diode) is about a volt positive (and noticeably "squashed") on the positive half-cycle of the input waveform. This is because conduction on the PA starts when the base is about one diode-drop above ground and doesn't vary much in voltage as the drive is further increased. Without the diode, the negative signal swing can be quite large- it's not unusual to see a few volts of negative signal swing. With the clamp diode in place, the negative swing is constrained to near ground, the DC average is pushed upward, and the base of Q6 is driven harder. For the curious, the presence of this added diode appeared to have no adverse effect on spectral purity.

Q: Paul (AA1MI) and I had a recent private discussion of the SW40+ circuitry involving Q5, and its collector load thru T4, the ferrite 8:1 transformer. We both agree that it may be useful to others. Glen VE3DNL

Original posting from Paul (AA1MI) to QRP-L:

I've been looking over and playing with the latest installment (RF buffer/driver) and have a few questions for our online sages...

1. The design consists of a 3-transistor output stage (Q4, Q5 and Q6). Why is it necessary to have three stages? Wouldn't just two suffice — one to buffer the mixer output/filter, in turn driving the PA transistor (here, Q6)?

2. I think I understand the purpose of T4 (improving gain on a Class A amp?), but I sure would appreciate it if somebody could walk me through the process of determin-

ing how many turns on what kind of core are needed — and *why*. Thanks, Paul, AA1MI

Response from Glen (VE3DNL)

1. Good question. Dave has designed the RF amp stages very conservatively. Good approach. Consider that these transistors are CHEAP!!! like about ten cents each. If adding one extra makes your design more stable or reproduceable then its money well spent. You COULD do it with two, but you'd have to push'em to the limit.

2. I went thru the design of this stage and found that Dave pushed the collector EXACTLY right...beautiful job. Its a class-A amp. To get max power out of it, you want the collector current to swing from its quiescent value (I recall its something like 20 - 30 ma) down close to zero ma., and up to twice quiescent. At the same time, collector voltage should swing from +12v down close to the emitter voltage, and up to +24 v on the overshoot. For both current and voltage swing to reach these limits simultaneously requires a specific collector load Z. That's where the turns-ratio of T4 comes in. If quiescent Ic = 25mA, and collector voltage swing is 10v (some is taken up by the emitter resistors), then Rc for best efficiency is 10v/.025 ohms, or 400 ohms.

With a turns ratio of 8:1, the impedance ratio is 64:1 This means that the base circuit must be around 400/64 ohms (about 6 ohms). This load is mighty non-linear and can't be measured easily. (and remember that these numbers are from memory, and might not be exact). To choose the turns ratio requires you to know what impedance Q6's base circuit presents to the 1-turn winding of T4.

So you've gotta choose a core that gives an inductive impedance of much greater than 6 ohms per turn^2. A rule of thumb is to design for between 5 to 10 times the 6-ohm load. So at 7MHz you want about 50 ohms minimum inductance. That's an inductance of 1uh per turn. Choose a core using the "AL" winding value (units of henries-per-turn-squared) sometimes they spec AL in mH/t^2 or nH/t^2. I'd expect that the FT37-43 core has an AL value exceeding 1uH/t^2 (more is OK, less isn't).

A few follow-up questions:

Q. Why do you say the load is highly non-linear? And how does somebody go about estimating it in order to size T4?

A. Q5 is the last linear stage: all the active devices beyond are "ON" during part of the (7MHz) cycle, and "OFF" during other parts of the cycle. This includes D6 and Q6, D7-D10. (D12 should never conduct in normal operation). When "ON", the base of Q6 is VERY low impedance. When "OFF" it is very high impedance. Trying to figure out the average impedance isn't easy. Same with D6. I suppose you could say that Q5 has an input Z that varies with instantaneous input signal, but it varies only slightly. When we say that Q5 is linear, we assume that input Z and output Z DON'T vary with input signal, and its a pretty fair approximation. We can't make the same approximation with D6 and Q6 because their input Z and output Z vary WILDLY with instantaneous input signal level.

Q. So you've gotta choose a core that gives an inductive impedance of much greater than 6 ohms per turn. Why? I only need 6 ohms, right, so one turn does the trick? A rule of thumb is to design for between 5 to 10 times the 6-ohm load. Rule of thumb for what? I'm not sure what you're referring to here...

A. Well, we don't want the inductive reactance of the transformer to load down Q5's collector. Q5 should be seeing mostly a resistive load, consisting of the (transformed) impedance of Q6's base circuit. Since T4's inductive reactance is in paral-

lel with Q5's collector, and in parallel with (transformed) load of Q6, it should be much larger than the resistive load that Q5 sees. This will assure that most of Q5's collector current goes into the load, and not into T4's inductive reactance. This is an important thing to see.

Q. So at 7MHz you want about 50 ohms minimum inductance. That's an inductance of 1uh per turn. How did you arrive at that figure (1 uH/turn)?

A. What inductance gives 50 ohms at 7 MHz? It is 50/(2*PI*7e6). That works out to a little more than 1e-6 henry (1uH). Since T4's secondary winding is 1 turn, then AL is 1uH/turn^2

Q. I noticed that many example circuits use a tuned tank at the output, putting a cap across the transformer. Why not here?

A. Yes, a tuned collector load could work. It'd be pretty low Q because of the low impedances involved. But still, the tuned circuit might require a trim-capacitor to resonate at 7MHz. Costs more, and its another tuning stage that builders can misalign. The broad-band transformer that Dave uses is cheap, requires no tuning, and has about the same performance.

Q. You say, "since T4's inductive reactance is in parallel with Q5's collector..." Hold on!! The transformer is in series with Q5's collector, not in parallel, right? Even looking at the Handbook's equivalent circuit for a transformer, all the elements are series (except for stray capacitances).

A. Didn't mean to add confusion. I've pulled out the only Handbook available,(1978) and am referring to their audio transformer modelling discussion... The Handbook talks about a few different loss mechanisms in their transformer: due to magnetizing current, leakage current, and winding resistance losses, and core losses. In the '78 book, they show the transformer model you've described:

It shows Rc (core losses) in parallel with the primary.

It shows Rp (primary wire resistance) in series

It shows Xp (leakage inductance loss) in series

Then it shows an ideal transformer labelled "PRI" and "SEC"

On the secondary side:

It shows Xs (leakage inductance loss) in series

It shows Rs (winding resistance loss) in series

Then the transformer model's output Es.

I looked for a more RF specific model that included some stray capacitances but didn't find any. In our application (T4) this model is OK because the coupling coefficient between primary and secondary is almost 1.0. However, the model shown doesn't discuss the path for magnetizing current. Since coupling is tight, Xp and Xs are small. And Rp and Rs are small too, since we have so few turns. Consider what happens in the extreme case where we disconnect the load. Secondary current goes to zero. Its a pretty good approximation to assume the whole secondary part of the transformer disappears. So you're left with Rc, Rp, Xp and the primary winding "PRI".

What isn't discussed in the '78 Handbook clearly is that in this model, some current still flows thru the primary side in this case. It flows around thru PRI, and is mostly inductive in nature (meaning that the primary voltage leads primary current by close-to-90-degrees). This is the magnetizing current. And we'd like this current to be small, compared to the primary current that flows when the load IS connected. To accomplish this, we need to have the inductance of PRI to be much larger than the transformed load resistance. This is where the "rule-of-thumb" of *5 to *10 comes in, to make magnetizing current 5 to 10 times lower than transformed

load current.

OK, now back to the T4 being in parallel with the collector. I consider that the collector output voltage is measured with-respect-to ground. For RF purposes, the top end of T4 is at ground too, because of the bypassing effect of C111 and C110. So in my mind, T4 is actually in parallel with Q5's collector. Most times, the supply voltage and ground are the same for AC signals. Make SURE you see this...its REALLY important.

Q. You then again say that T4's reactance is also "...in parallel with (transformed) load of Q6..." Again, I don't see any parallel circuit here; the secondary of T4 is in series with the base of Q6, which presents the load.

A. The secondary winding of T4 is in PARALLEL with its load. I think we'd both agree that D6 is in parallel with R29. Can you see that the base of Q6 is in parallel as well? If so, then you'll have to agree that the secondary winding of T4 is in parallel as well. I'm mentally shorting out C35, because its a very low impedance for RF.

Q: My interest has been "peaked" while reading the first chapter of "The Art of Electronics," a text recommended on QRP-L. I have been reading a section on Thevin's equivalent circuit which the author's use as a basis for understanding impedance matching between circuits.

A. Thevenin and Norton equivalents are VERY powerful analytical tools. Used extensively to simplifiy linear circuits. Learn 'em well.

Q. What are the input-output impedances associated with the circuitry associated with Q4 and Q5? I would appreciate the logic supporting the answer. I am comfortable with only resistance and DC circuits at this time, but I am not quite certain how to incorporate active devices and reactance components in the computations.

A. You need to learn a little more about active circuits before you can work out Q4's input Z. But you can find an upper limit by calculating the equivalent resistance of Q4's bias resistors, R23(22K) and R24(10K). You should be able to see that Q4's base will be in parallel with these resistors, and can only reduce the result of the above calculation. Since we're dealing with AC signals, consider the top of R23 to be at AC ground potential. So for AC, R23 and R24 are in parallel.

Q. C114 by-passes only a part of the Q5 emitter resistance. Why? I "thought" good practice dictated a by-passing of the total emitter resistance was necessary to assure a "stable" circuit, that is, to avoid oscillation. Obviously, I have missed something in my past study/reading.

A. Actually, the circuit is more stable with some of the emitter resistance un-by-passed. The gain is lower as a result. And Q5's input impedance is higher too. Q5 becomes a more linear amplifier as well. A good, conservative design.

Q. The directions for peaking T2 and T3 do not exclude the use of a metallic tool. I used both a metal small screwdriver and a homebrewed plastic "screwdriver" with no discernible difference in adjustment. However, I have some other kit building experiences with the adjustment of coil/capacitor/ferrite which required the use of a plastic tuning tool. What gives?

A. T2 and T3 are cup-cores. The screwdriver slot is mostly out of the flux path. This design is mostly self-shielding. And T2, T3 tune fairly broadly too, so you don't notice much de-tuning. I'd still use a plastic tool though. A slug-tuned threaded core with a hex-hole is a different story. Here, the flux-path is terribly distorted by a metal tool.

Here is the short quiz from part 6 with an answer. Most of those who responded had the correct answer. Some did not, but I

think it was because they did not read the question carefully.

Q. Now while trying to peak T2 and T3 again I accidently bumped the ground shield on my scope probe to the top part of R20 (the side not connected to V+). Can anyone guess which component I fried when I keyed up, and why? (This is homework.)

A. 12V (or 13.8) from emitter to base of Q3 would fry the junction - almost certainly open, unless your supply has a quick current fold-back, in which case the junction might wind up shorted. Not likely though. Also - it wouldn't matter if the rig was keyed or not.

Scope waveforms of the SW-40+

Some oscilloscope waveforms of the keying process have been taken to show how the radio switches from receive-to-transmit and back. The oscilloscope is particularly useful here. Waveforms of the final amplifier base and collector voltage are shown as well.

Equipment setup:

A TDS210 digital oscilloscope with two x10 attenuator probes was used to capture the waveforms. A function generator was used to "key" the rig in a predictable way: the transmitter was activated for a time period of 0.18 seconds, and then the receiver was allowed to operate for another 0.80 seconds. Keying was repeated continuously this way. Antenna connection went to a 50-ohm dummy load. No headphone was connected (open-circuit). DC power came from a regulated bench supply at 12.3V.

For the keying waveforms, the 'scope was triggered "externally", directly from the function generator that was keying the radio. An analog 'scope usually displays waveforms that start the sweep at the left coincident with the triggering event. With a digital 'scope like the TDS210, you can see stuff that happens *before* the trigger event. So some waveforms would be very difficult to duplicate exactly if you're using an older-style scope.

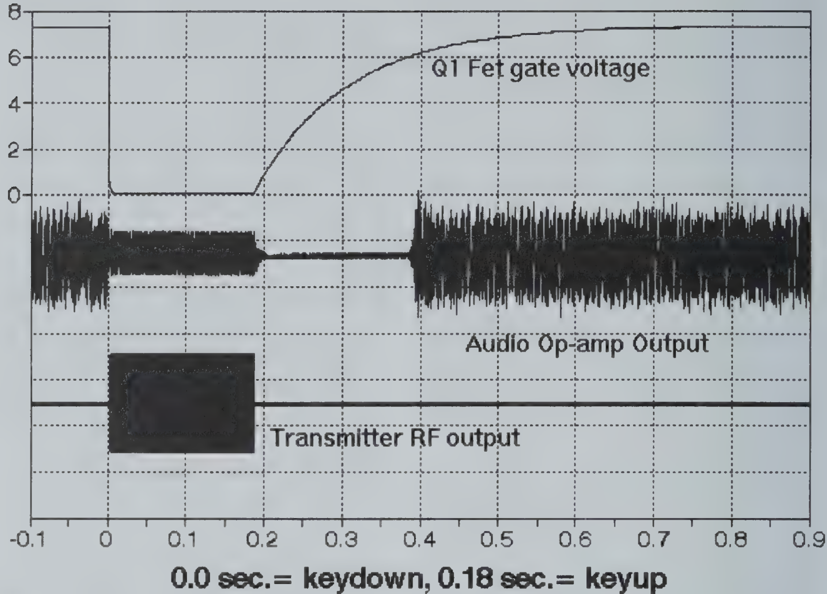
Oscilloscope waveform data was transferred to a PC computer over a serial port. Data was imported to a spreadsheet where it was graphed. WINDOWS **Paintbrush** was used to cut-n-paste some graphs together, and annotate and title them.

Waveforms explained, scope triggering

The first graph shows one keying event, lasting 0.18 seconds. The trace actually starts at the left 0.10 seconds BEFORE the key activates. The timescale is shown so that 0.0 seconds corresponds to the moment when the key is activated.

If you're setting up a 'scope to view these waveforms, you might want to connect one channel to the keyline, say channel 1. You'd set the 'scope to trigger from channel 1, negative slope (falling edge), and DC coupled. Many scopes will require that you switch to "normal" triggering rather than "auto" at these slow sweep speeds. Adjust the trigger level while keying the radio till you see that hitting the key initiates a sweep. The scope screen should be blank if you don't key the rig. Now you can use the other channel (channel 2) to probe around the radio and view various parts of the radio that only change when keying is activated. Of interest are the FET gate (Q1), the transmit supply switch (Q3), RF amplifier (Q4, Q5) and perhaps the transmit mixer (U5).

During the first 0.1s at the left, you can see audio (noise) output from U4, pin 7 up until the key is hit. The sweep here is so slow that you can't see the waveform going up and down, but you can see its "noisy" envelope. Very soon after the key is activated, the FET switch opens up, and the rig starts transmitting. During this time, audio consists of a constant-amplitude sidetone of about 800 Hz. When the key is



These waveforms represent Q1's gate voltage (top), the audio output at pin 7 of U4 (middle) and the 7 MHz. RF transmitter output at the antenna (bottom).

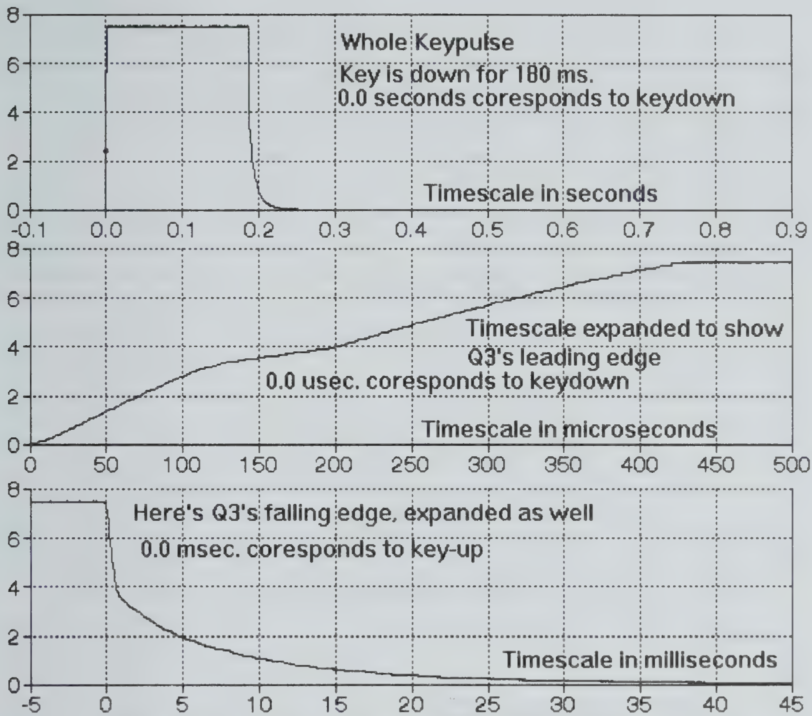
released, both the transmitted waveform and the sidetone stop.

After that, the audio output is silent for a while, till the FET gate charges back up. Then the FET switches back ON and audio noise re-appears for the rest of the trace. The keyline not only opens up the FET gate, but it turns ON the transmitter amplifer supply via Q3. The next set of three traces explores the keying timing involved with raising, and lowering the supply to U5. Voltage rises up to 7.5 volts, where zener diode D11 conducts. Q3's collector continues all the way up to +12v.

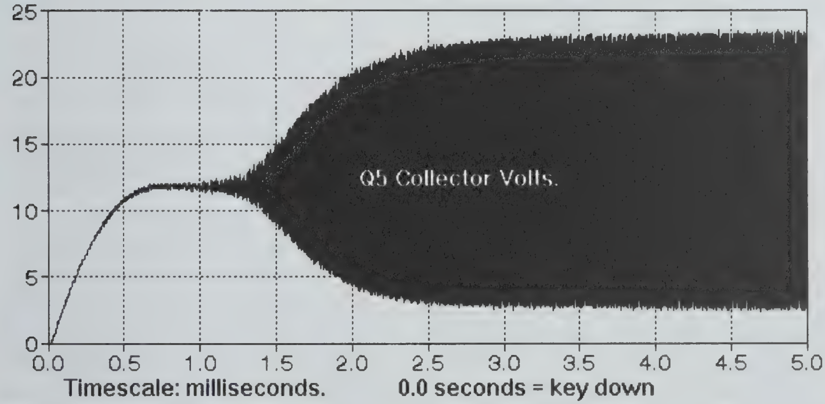
The top trace shows U5's supply at the same timescale as the previous traces: key is activated at 0.0s and de-activated at 0.18s. Q3 pulls up to nearly 12v very quickly after the key it activated. The middle trace examines its rising edge in more detail - notice that the sweepspeed is much faster than the top trace, and that the keydown event occurs at the extreme left edge of the sweep. U5's supply ramps up to 7.5 volts about 430 microseconds after the key is activated.

The bottom trace examines Q3's falling edge, showing how it falls from 7.5v back to zero. Here, 0.0 seconds corresponds to the key-up event. Q3 actually turns off quickly, but charge on C110 takes time to bleed away.

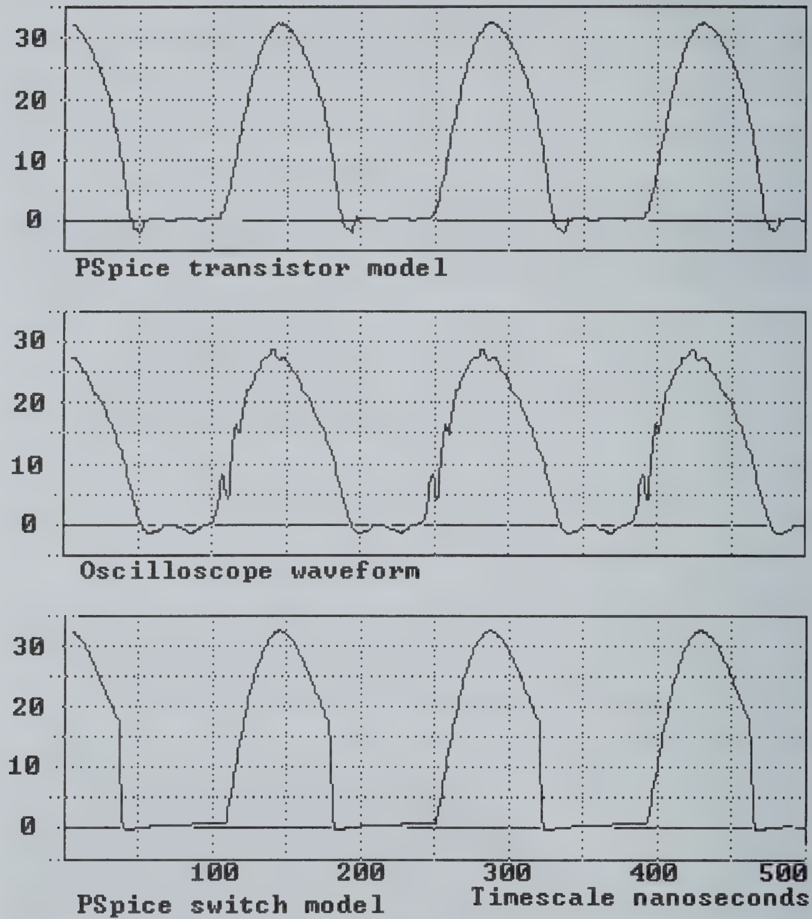
Well, its fine to see the supply voltage going up and down, but does the RF waveform follow it in a similar fashion? No, it lags a bit, as the next trace shows. Q5's collector voltage is shown as the key is activated. You can see that it exponentially rises up to +12v when Q3 turns on, but RF doesn't come out immediately



Q3 collector voltage shown at three different timescales



The RF envelope (at 7 MHz) starts to come out about 1 millisecond after keying, and only rises to full amplitude after about 3 milliseconds. Although U5's supply is up to 7.5v after only 0.43 ms., its 4 MHz. crystal oscillator (connected to pin 6, and pin 7) takes some time to build up. U5's output at 7 MHz. depends on the oscillator, and doesn't come up right away. The laggard RF goes thru the buffer at Q4 and into Q5.



The last waveform here hasn't anything to do with keying. It shows the 7 MHz. voltage waveforms around Q6, the final RF amplifier. RF drive was a little high, so that two watts output was sent to the 50-ohm dummy load. The collector waveform's average voltage is +12v. It swings down to ground (Q6 is conducting a lot of current at this time). But when Q6 is not conducting, its collector voltage swings high above the supply voltage on the rebound - up to 28v. This transistor is working somewhat beyond class C, since it is saturated when ON.

Base voltage is limited by D6 on the negative swing, and by the base-emiter of Q6 on its positive swing. Its the positive swing that turns Q6 ON - while the base voltage swings negative, Q6 is OFF

Part 7.

In the transmit chain we left off without installing the final amplifier transistor. This was done on purpose. With the final out we don't have to worry about frying the final because we forgot to connect the dummy load. We will install it last before

the final calibration.

The reason for building the transmit circuit first was to provide us with a signal source for testing the different sections of the receiver. This should make trouble-shooting the receiver easier for those who don't have an RF signal generator.

Gather the following parts:
T1 IF can (the last one in the kit)
C1 47 pF
C40 47 pF
C101 .1 uF
D7 1N4148 diode
D8 1N4148
D9 1N4148
D10 1n4148
RFC3 10uH inductor (brn blk blk) in the "misc" bag of parts
8 pin socket for U1
U1 NE612 mixer IC

Install the above components using the same caution as in the previous lessons. CHECK YOUR WORK!

Now we are going to cheat a little. We are going to "borrow" some signal from the output of the transmit mixer so that we can get a nice strong signal for the receive sections.

Two TEMPORARY jumpers are needed for this. I used two of the trimmed leads from one of the capacitors that I just installed. Solder them lightly, as they will need to be removed later. I soldered mine on the top of the board. All references are looking at the front of the board, with T1 in the lower left hand corner. Connect the first jumper from J1 between pins 2 and 3 (Those are the top two). The second jumper is from the base hole for Q6 (left most hole) and and the top hole for C36. Remember that all references to "top" and "left" are with you looking at the top of the board with T1 at the lower left hand corner.

What these jumpers do is bypass Q6 (which is not installed yet) and bypass the RF Gain potentiometer. This feeds the transmit signal from the base of Q6 into the input of T1.

Set T1 to mid range. This will be tweeked later. Turn on the rig and check for smoke ;-) None? Good. Ok, put your RF volt meter or o-scope on pin 5 of U1. An easy place to measure this is at the top pad of C11. The o-scope users will see a little (millivolt) signal leaking through from the VFO. Key up the rig. You should see RF at pin 5. O-scope users should see a complex waveform at pin 5. Mine was several volts peak to peak. This waveform should look similar to the one we saw at the output of U5, the transmit mixer, before the bandpass filter.

Here is a little theory. From the antenna the receive signal first passes through the first bandpass filter consisting of L3, L4, C37-39 (not installed). This cleans up the transmitted signal during transmitting and preselects the 7Mhz band during receive. The signal is coupled through C40 to the four diodes whos function is to limit the signal reaching the receiver by clamping it to ground if it is over 1.4 volts. The only time that it will reach that level should be during transmit. The signal from the transmit that reaches the receiver provides sidetone during key down.

The RF gain pot (not installed) is used to reduce the RF from the antenna while receiving a strong signal. The connection of this part is a little unusual and it took me a while to figure it out. In other instances where control of a signal is required, you see the signal injected into the "top" of the pot, the bottom of the pot grounded, and the output signal taken from the wiper. The problem with this setup is that as the wiper postion is changed the output impedance is changed also. In Dave's configuration the impedance is relativly steady across the wiper postions. This is important as changes in this impedance will effect the resonant circuit of

T1.

T1, which is set to resonance at the receive frequency, couples the RF into U1. Here it is mixed with the VFO signal and is available at pin 5. This signal will contain two major components and several minor components. The major components are RF+VFO and RF-VFO. At 7 Mhz with a 3 Mhz VFO, there will be 10 Mhz and 4 Mhz. The 4 Mhz is the one that we are interested in as it is our IF frequency. All these unwanted "nasties" will be removed by the next section (part 8) the crystal filter. More detailed info on the mixer can be found in part 5, the transmit mixer.

Remember that our transmit mixer oscillator (Y5) was "pulled" lower in frequency by RFC2 and C29. This gives us the proper offset for receiving on the proper side of the signal. At a receive frequency of 7.040 Mhz the VFO will be at 3.040 Mhz and the transmit frequency should be about 800Hz lower, or 7.0392 Mhz.

SW40+ Receiver Front End

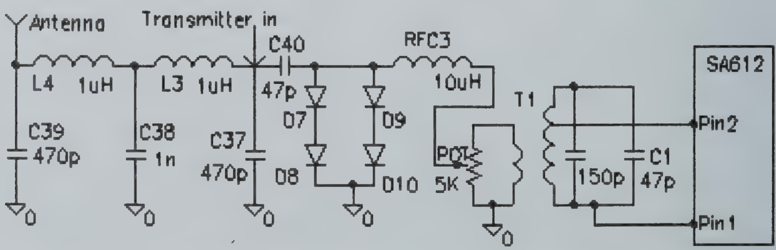
This note describes the circuits between antenna and mixer U1(SA612). Circuit operation during receive as well as during transmit is described since the very large transmit signal modifies circuit operation. The circuitry is divided into functional blocks: a good way of separating function for debugging. Each module serves a different purpose.

PI Filter: The first circuit "module" is the five-element low-pass filter involving **C39**(470pF), **L4**(1uH), **C38**(1000pF), **L3**(1uH), and **C37**(470pF). You should see that this circuit is symmetrical: it looks the same from either end. Indeed, the transmitter propagates its signal from right-to-left out to the antenna, while the receiver uses the same circuit in the other direction. In any case, it is a low-Q filter that attenuates signals above 7MHz, while passing all below 7MHz. Ideally, this filter should be terminated in a 50-ohm resistance. Due to its low-Q, this requirement is somewhat relaxed, and we shall see that the receiver doesn't comply here. Circuit function is more appropriate to the transmitter, to attenuate harmonic output. But why not take advantage of its filtering action for the receiver?

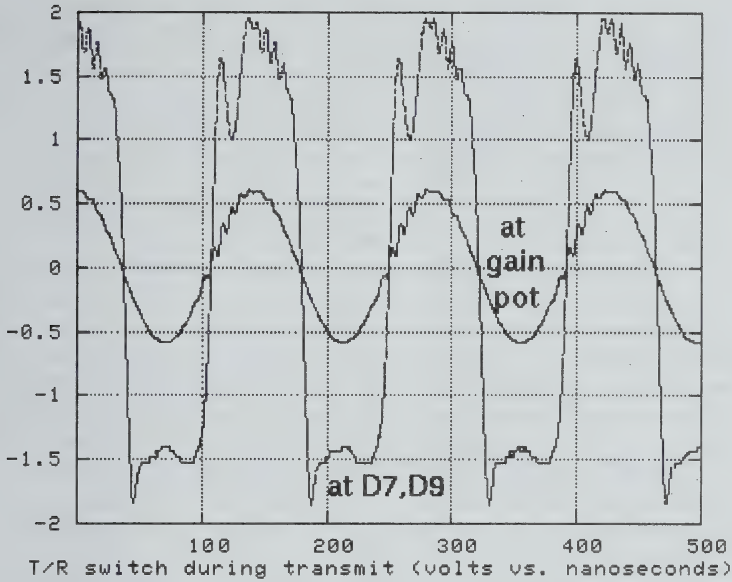
T/R switch

The next circuit "module" blocks most of the transmitter power from overpowering (and destroying) **U1**. We want most of the transmitted power to go out the antenna, not into the receiver. Yet we want *all* the received signal to pass through this circuit with little attenuation, into **U1**. Circuit components here include **C40**(47pF), **D7 - D10**, and **RFC3**(10uH).

Consider what happens while transmitting. A very large amplitude waveshape (not a sinewave) appears at **C40.** This point is common to both receiver and transmitter. That is, when not transmitting, the very small received signal must pass this



SW 40+ Receiver Front End Schematic



same point on its way into **U1**.

The transmitted signal is of such large amplitude, that **D7- D10** conduct 7MHz. current to ground. Current is limited by the impedance of **C40**...480 ohms at 7MHz. While conducting, these diodes look like low-impedance to ground. However, their forward-bias voltage of 0.6v each mean that there is still a 7MHz. signal across them. But instead of 30v p-p it is clipped to about 3.3v p-p ("**at gain pot**"scope trace). The **RFC3** choke further attenuates the transmitted signal so that the **R.F. GAIN** pot sees even less ("**at D7,D9**" scope trace). However, from there, the voltage amplitude is boosted by **T1** (tuned to 7 MHz.) on its way into the mixer due to its step-up turns ratio.

The critical parameter of the mixer's input transistors is their reverse breakdown voltage between base and emitter. For R.F. transistors, this voltage limit is about three to five volts. Anything less won't cause damage.

While receiving, the transmitter is idle, and looks like a very small capacitance in parallel with **C37**. At 7 MHz., antenna signals pass through the PI filter unmodified and appear at **C40** with no attenuation. Since the amplitude of received signals is so small, **D7-D10** do not conduct, and they appear as a tiny (insignificant) capacitance to ground. This allows **C40** and **RFC3** to work in concert as a series-tuned circuit resonant at 7 MHz.

The series resonant circuit sees about 50 ohms on its antenna side (**C40)** and about 100 ohms on the mixer side (**RFC3**). So its loaded Q is low: about 3. For such a low Q, exact trimming is unnecessary: resonance is "close enough" to 7MHz. At this frequency, reactance of **C40** and **RFC3** cancel, leaving an almost direct connection between **C37** and the **R.F. Gain** pot (and to the link winding of **T1** too).

You might ask how large an antenna signal would have to be before **D7-D10** began conducting? If amplitude across each diode rises to perhaps 0.6v, diode resistance may start to load the series-tuned

circuit of **C40/RFC3**. A Q of three means that the diodes see three times the antenna amplitude. So that reduces the diode conduction limit to 0.2v. With two diodes in series, maximum peak voltage would double to 0.4v, and we could withstand a 0.8v p-p antenna signal before conduction begins.

7MHz. tuned Transformer (T1)

The PI filter, and the series-tuned T/R switch provide inadequate selectivity to reject unwanted mixer input signals (particularly at the image of 1.0Mhz). **T1** is a tuned bandpass filter that attenuates signals both above and below the 40M band. In addition, it also steps up the low antenna impedance to more closely match mixer input resistance of 3000 ohms. Here are the turns ratio of windings and taps:

· Link winding: 2 turns

· Tuned winding: 17 turns, tapped 11 turns up

Let's see how the mixer impedance matching works out... First, what's the **SA612** input impedance? The data sheet says "unbalanced: 1500 ohms". However you should note that neither input (pin 1 nor pin 2) are bypassed to ground. So the input tends to be balanced at twice this value, and the 11-turn winding section of **T1** sees 3000 ohms. Assuming that flux linkage within **T1** between the 2-turn link and the 11-turn winding is 100%, then the impedance ratio is $(11/2)^2$ or 30.25. So the 3000-ohm mixer load looks like 99 ohms at the link winding.

Since the series-tuned T/R switch has nearly zero impedance at 7MHz., the PI filter sees 99 ohms as well. And so does the antenna. Measuring with a noise bridge verifies that at **T1**'s 7MHz. resonance, input Z is indeed 100 ohms. Now this is not a very good match to a 50-ohm antenna system. Why did Dave not use the full 17-turn winding instead? This would give about a 41-ohm impedance at the link...a closer match to a 50-ohm system.

There are a number of possible reasons:

· The full 17-turn could present too high peak voltage to the SA612's input transistors during transmit

· The loaded Q of T1 would be lower, resulting in poor image and spurious rejection

· Overload (intermodulation IP3) is worse, resulting from bigger input voltages.

As a matter of fact, you could improve the robustness of the front-end by changing to the 6-turn link section rather than the 11-turn section: connect **SA612's** pin 1 to the other end of the transformer (requires cutting a P.C. trace). The price you pay is poorer sensitivity: if you have a good antenna, it is a reasonable price to pay.

Part 7 Questions and Answers:

Q. C40 and RFC3 form a series resonant circuit at the input of the receiver. My calculations show that it is resonant at 7.34 Mhz. This is outside the 40 meter cw band. How does this effect receiver performance?

A. But did you calculate the Q? Very low. There's about 50 ohms at the antenna-side. And on the receiver side, at LEAST 50 ohms more. So Q is MAXIMUM 5. That covers the whole band. And if there IS any left-over reactance, T1 can tune some of it out.

Q. The diodes D7-D10 are meant to limit input signal. Why are they in the middle of the series resonant circuit made up of C40 and RFC3?

A. If the diodes weren't there, the voltage at this point would be Q6's collector voltage (25v p-p) MULTIPLIED by the Q calculated above And U1 (or the RFgain pot) would certainly get fried by all that QRP power. Not only do the diodes clip it down to about 2.6vp-p, but they destroy the Q of the series-resonant circuit of C40-RFC3, since while conducting, they're very low-

Z devices. So while transmitting, RFC3 is acting as a proper choke, with 440 ohms inductive reactance in series with the receiver. But while receiving, this reactance is tuned out by C40, because the diodes are NOT conducting.

Q. Explain what is happening impedance-wise with the RF gain pot, T1 and U1. Why is C1 150 pf instead of 47 pf like in the transmit mixer?

A. C1 is a goof. It IS 47pf. T1 is tuned to 7Mhz. Don't know what the turns-ratio is, but i'll bet it steps up the 50 ohm antenna Z to something higher for U1's input of 1500 ohms.

Q. Why is the signal for U1 taken from the center tap of T1, not across the whole thing? I thought we would need as much signal as possible from the antenna to make it to the receive mixer.

A. I intend to find the turns ratio of T1 (don't know it yet). The tap could be anywhere, not necessarily at center. Using the tap instead of the top means the loaded Q of T1 is high. These canned tuned transformers usually have unloaded Q's approaching 100. With only ONE tuned circuit at the front end, we need a high Q to reject spurs, and images. Tapping U1 down maintains the Q reasonably high. Its also a good idea to tap-down too low rather than too high. This'll improve U1's large-signal handling.

You had some good questions about T1. I just measured its turns ratio (roughly). If you take the full tuned winding length as 100%: the link is about 13% the tap-point is at about 68%. So if U1's input Z is 1500 ohms, then the turns ratio of (.68/.13)^2 transforms down to about 54 ohms. Not a bad match to the antenna. Loaded Q isn't very high. Something like 10

Part 8

Gather the following components:

C11 47pF
C12 150pF
C13 150pF
C14 150pF
C15 150pF
C104 .01uF
R1 470 ohm
RFC1 22 microhenry
Y1 4 Mhz crystal
Y2 4 Mhz crystal
Y3 4 Mhz crystal

Install the above components into the board. Use caution while installing the crystals Y1-Y3. If they are mounted flush to the board the cases could short out the solder pads underneath. I mounted mine about 1mm above the board surface using a temporary spacer while soldering. I know that you can get insulators that fit over the bottom of the crystals but I had none. Use whatever method you would like to prevent the crystal cases from touching the solder pads on the top of the board.

You can ground the cases of the crystals now by soldering a solid wire across the top of the 3 crystals. DON'T OVERHEAT THE CRYSTAL CASES AS DAMAGE CAN RESULT! Now attach the wire to the solder pad on the left side of the crystals. It should be the only open solder hole on the left side of the 3 crystals. When you are done you should see a single wire going from the right most crystal, soldered across the top of all 3 crystals, then down to the solder pad on the left side. This grounds the cases of the crystals preventing interferance from strong stations from getting past the filter.

Ok, check your soldering, parts placement and parts value. Got it all right? Good. Now apply power and check for smoke. Place your scope probe or RF probe on the solder pad for U3 (not installed yet). Key up the rig with your temporary jumper. You should see a nice pretty 4Mhz sine wave. On my scope it was about 0.5 volts peak to peak. That should be enough of a

voltage to tickle those RF probes. What happened to all those nasty mixer components that we had at the output of U1? Well, thanks to the crystal filter they are all gone!

So now you have to ask the simple question "how does it work?" For me, this part of the circuit is shrouded in mystery. Please try to stay with me and lets see if we can get some discussion going on the subject of crystal filters.

Remember, the basis of a superhet receiver is that it converts the desired incoming signal to some "standard" frequency. In our radio that frequency happens to be 4Mhz. This is done because it is easier to design a filter that operates on one specific frequency than one that operates over a range of frequencies. It also allows you to reject the other sideband, unlike Direct conversion receivers. So the purpose of the receive mixer is to convert the desired receive signal to the IF frequency (4Mhz). The crystal filters job is to make sure that only the IF frequency makes it from the receive mixer to the product detector.

In the last section we completed the receive mixer that was centered around U1. This mixed our incoming signal from the antenna with the VFO and presented the mixed signals at it's output. Let's pick some round numbers.... Receive signal=7Mhz VFO=3Mhz The output of the mixer will contain many mixer products, the major ones being 4Mhz and 10Mhz. This circuit will filter out all but the 4Mhz signal.

I did a bunch of reading on crystal filters and came away even more confused then when I started. Most of the descriptions that I read either were emperical designs, or the design was referenced as being covered by one of the books in the bibliography. "Just build it this way and it should work" was a common phrase in these writings.

Here is what I do know. The crystals must be closely matched, within 10 to 20 Hz of each other. Dave at Small Wonder Labs was kind enough to do this for us. If the crystals are not matched the performance will be poor. All of the capacitors are the same, 150pf. The value of the capacitors has an effect on the bandwidth of the filter. It is important to match the input and output impedance of the filter to minimize the loss. I have no earthly idea on how to calculate this, but I do know what parts in the circuit are performing the impedance matching. C11 and RFC1 form an "L" filter to match the output impedance of U1 with the input impedance of the filter. The filter is terminated into R1 (470 ohms) before reaching U3.

The crystal filter is the core of nearly every QRP rig out there. I think it is important to understand what is happening here. Lets start a discussion on the function and characteristics of this filter and see what we can learn.

Mike,

I can sympathize. But handwaving explanations don't hack it when describing these multi-coupled circuits. So where does that leave us? It leaves us with the "black-box" situation that you describe. Don't knock black-boxes too much - it's a very powerful analytical tool in breaking down circuits. Xtal-filter is just one of those black-boxes where peeking inside reveals a lotta nasty math. Do you really wanna know where all those coupling coefficients come from?

Me - I'm math-adverse. After peeking inside, I'm content to use the design-tables But it IS useful to play about: What happens if you design for a wider bandwidth? Coupling caps go down. Terminations go up. What happens if you use low-Q crystals? In-band attenuation goes up. Passband edges become rounded. What happens if your termina-

tions are too low-Z? Passband ripple looks like a rollercoaster. What happens when your terminations are reactive? Passband gets skewed. SPICE simulations can help a lot to show this kinda stuff: use the book designs as a starting point, then play-about with SPICE. - Glen VE3DNL

Q. The thing I am having trouble with is this - why doesn't RFC1 effect the filter's shape and why doesn't C12 effect the impedance matching in this "swapped" position? Or is it that they do, but not significantly? I am intrigued by this because, as a neophyte designer, it isn't intuitive to me when one can "get away" with this.

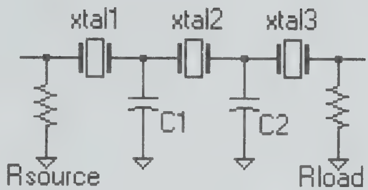
A. These components DO affect impedance match (and therefore filter shape as well). RFC1 and C11 form a "L-network" to match U1's output Z of 1500 ohms down to the filter's required impedance of about 350 ohms. However C12 is in there to modify Y1's series-resonant-frequency up a little higher (by 106 Hz) than Y2. It is not part of the impedance matching network.

We'll look through xtal filter design - Glen, VE3DNL

SW40+ I.F. Crystal Filter

This note proceeds through the design of a Cohn-type crystal ladder filter. Starting with fixed crystal specs and desired bandwidth, a simple generic filter is designed. The generic filter is then modified to accommodate source and load impedances found in the SW40+ receiver. Component values work out close to those in the rig.

The simplest form of the ladder-type Cohn filter requires at least two crystals, but any number can be cascaded. Between pairs of crystals and ground must appear a coupling reactance. Source resistance and load resistance must also be defined. A two-crystal filter would result in inadequate sideband suppression. Four or more crystals require extreme care in selecting



crystal frequencies, coupling capacitors and termination resistances.

Crystal Model

From the Handbook you should review the electrical equivalent of a quartz crystal. By far, the two most important crystal parameters are the motional inductance Ls and motional capacitance Cs. From my junk-box a number of 4.0 Mhz. HC-18 microprocessor crystals were characterized. These should be *hopefully* similar to those in the SW40+ kit:

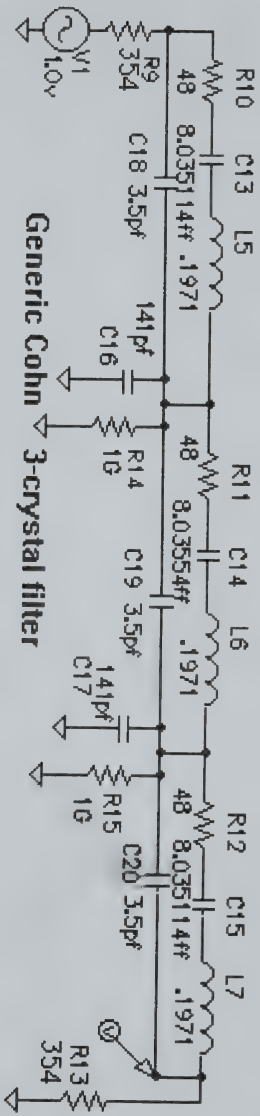
- Fs = 3999165 Hz. Series resonant frequency
- Ls = 0.1971 H Motional Inductance
- Cs = 8.03554x10-15 F Motional Capacitance
- Rs = 48 ohms Motional Resistance
- Cp = 3.5 pF Parallel plate capacitance

Besides crystal parameters, we must choose a desired filter passband width (BW). I choose 300 Hz. because I know from experience that component values will come out about right.

We can also choose from Butterworth, Chebychev, Gaussian, or Bessel responses. Each is optimized for different applications. One has minimum ringing, another has best shape factor. Butterworth response is reasonable for listening to CW.

Generic Ladder Design

I have followed the ladder design process as described in Handbook of Filter Synthesis by Zverev[2]. His ladder designs are exhaustively complete and unfortunately complex. The result is shown in the schematic "Generic Cohn 3-crystal Filter"



Generic Cohn 3-crystal filter

N	q	k_{12}	k_{23}	k_{34}	k_{45}
2	1.4141	0.70711	-	-	-
3	1.0	0.70711	0.70711	-	-
4	0.7654	0.8409	0.4512	0.8409	-
5	0.618	1.0	0.5559	0.5559	1.0

below. All Butterworth filters share coupling coefficients which determine the reactances of C1 and C2 of the generic filter above. For a Butterworth response, Rsource and Rload are scaled by another coefficient called "q" which is related to loaded filter Q (Fs / BW). Here is a table of Butterworth coefficients for ladder filters of N crystals:

For our 3-crystal generic filter, both coupling capacitors (C1 and C2) will have the same value, because k_{12} and k_{23} have the value of 0.70711. Their capacitance is roughly:

$C1 = Cm \times Fo / (BW \times k_{12}) = 8.03554 \times 10^{-15} \times 4000000 / (300 \times 0.70711) = 152$ pF

$C2 = Cm \times Fo / (BW \times k_{23}) = 8.03554 \times 10^{-15} \times 4000000 / (300 \times 0.70711) = 152$ pF

Since each of these capacitors sees Cp from two adjacent crystals, we should subtract off 2 x Cp from these values leaving us with 145pF. In any case, these are close to the values Dave chose for C13 and C14 in the SW40+ schematic.

Now let's try a simplified equation for Rsource and Rload. Our generic filter is symmetrical, so these will have the same value:

Rend = 2PI x Ls x BW / q = 6.2832 x 0.1971 x 300 / 1.0 = 372 ohms

These simplified values are close to those derived from the complex methods of Zverev[2]. PSPICE's requirement that all nodes not "float". There is one more aspect of this filter that we haven't addressed: frequency matching of the three crystals. For proper tuning, the two end crystals should have a series-resonant frequency 106 Hz. higher than the center crystal (from Zverev's design process). In the Generic Cohn filter shown, C13 and C15 have been decreased to reflect this frequency offset. In practice, we cannot modify our sealed crystals, and must execute the offset a different way so that three crystals of identical Fo can be used.

The frequency response of this filter is plotted below (Generic). Were the filter lossless, output voltage would be 500 mv.

This generic filter will now be adapted to make it work using three identical crystals. Since the SA612 source and load resistances are different from the generic design, matching networks must be added as well.

Compensating for 106 Hz. Offset

We can raise the series resonant frequency of the two end crystals by adding a capacitor in series with each. Currently, series resonance is:

$Fs = 1 / (2 * PI * SQRT(0.1971 * 8.03554 \times 10^{-15})) = 3999164.7$ Hz.

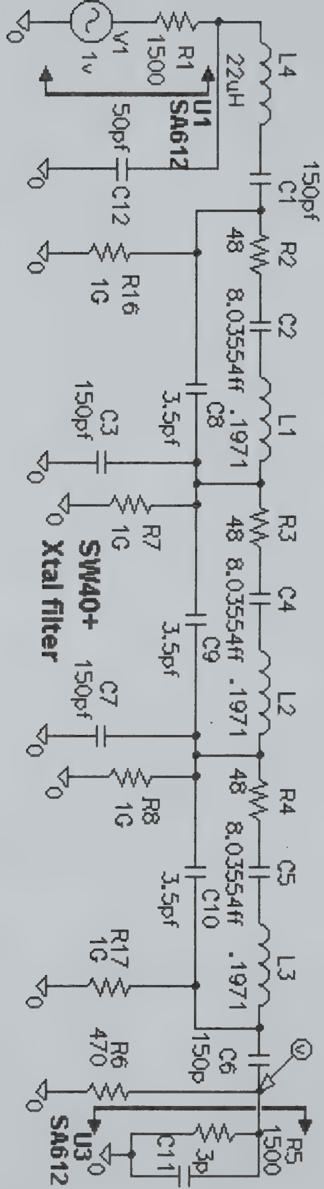
What capacitance would raise Fs to (Fs + 106) ?

$Cs(new) = 1 / ((2PI \times (Fs+106))^2 \times 0.1971) = 8.035114 \times 10^{-15}$ farad

Cs(new) is the total capacitance of two in series: our original crystal motional capacitance (Cs) of 8.03554 ff and our externally added modifying capacitor. Knowing Cs and Cs(new), we can find the value of the external capacitor:

Cexternal = (Cs * Cs(new)) / (Cs - Cs(new)) = 151.56 pF.

By adding this capacitance in series with Xtal 1 and Xtal 3, the 106 Hz. offset is accommodated - and all three crystals can be frequency-matched to within about 30 Hz. These capacitors appear in Dave's SW40+ schematic as C12 and C15. Is it coincidental that the coupling capacitors are the same as these frequency-tuning



SW40+
Xtal filter

capacitors? For this filter (and *only* for 3-crystal types) this will always be the case.

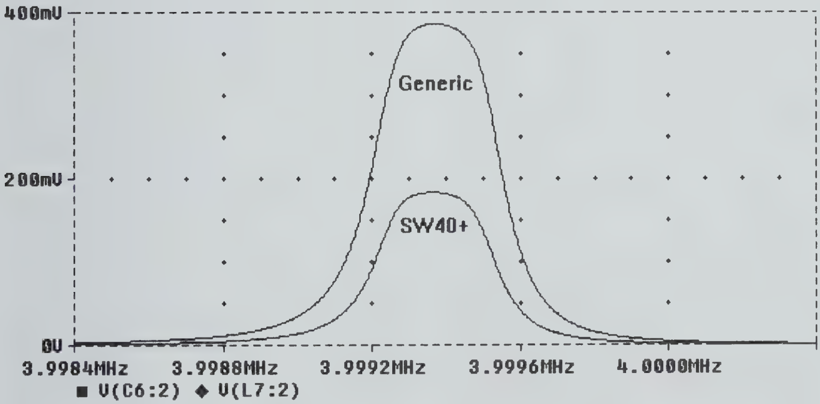
Matching to SA612 source and load

Driving the filter is a SA612 chip (U1), with 1500 ohm source impedance. The filters' load is also a SA612 (U3) whose input impedance is also 1500 ohms. Our filter would like to see terminations of 354 ohms.

Dave has chosen to simply load down the filter's output with a 470 ohm resistor (R1). This resistor, in parallel with the chip's 1500 ohms works out very close to the required 354 ohms. At the filter's input, Dave has chosen to add an L-matching network consisting of a 22uH choke (RFC1) and a 47pf capacitor (C11). Let's see how well these component values do in matching the required 354 ohms to SA612's 1500 ohm source:

At 4 MHz., the choke's reactance is +j553 ohms. C11's reactance (added to the SA612's 3pf output capacitance) is -j796 ohms. What's the parallel equivalent of 354+j553? It is 1218 ohms in parallel with inductive reactance of 780 ohms. C11's reactance nulls out this equivalent parallel reactance quite well, leaving 1218 ohms resistance - nearly matching to SA612's 1500 ohms. Here's the PSPICE circuit showing the SA612 equivalents at both ends of the filter:

This filter's response is overlaid ("SW40+") in the frequency plot above. Let's take a quick look at filter attenuation. Each crystal has a series resistance of 48 ohms. For the generic filter, if we assume that all filter reactances cancel somewhere close to the filter's center frequency, we have left 3 * 48 ohms series resistance dumping signal into a 354 ohm load. That's a simple voltage divider that delivers 0.713 of input voltage to the output.



For the SW40+ filter, additional losses are incurred because of the 470 ohm loading at the output. That's why the amplitude response is lower than the generic filter. The input side is matched, the output isn't.

References:

1 Hayward, Wes A Unified Approach to the Design of Crystal Ladder Filters QST May, 1982.

2 Zverev A.I. Handbook of Filter Synthesis Wiley 1967.

Part 9

The section we are going to build next is the product detector. Remember where we last left off, we had mixed the receive signal with the VFO down to the IF frequency. From there it is "magically" filtered by the IF crystal filter. Now we want to take that signal and turn it into something we can hear with our ears. This is the job of the product detector. It mixes the filtered IF with it's own oscillator that is running at the IF frequency. In this case, if the VFO frequency is 3Mhz, the transmit frequency is 6.9992Mhz (VFO + xmit mixer freq 3.9992Mhz). Remember that the crystal frequency in the oscillator of U5 is pulled 800 Hz lower by the addition of RFC2. This provides the proper xmit signal offset of 800Hz on the upper sideband. Remember that we are using a small part of the transmit signal as our receiver source, so the receive signal frequency is 6.9992Mhz. Once it is mixed with the VFO we get VFO - receive=3.9992Mhz (look familiar?). This is within the 500 hz bandwidth of the IF filter so this 3.9992 Mhz signal is presented to the product detector. The product detector mixes this signal with its own internal oscillator running at the IF frequency of 4Mhz. The resulting signals are 7.9992Mhz and 800Hz. The 800Hz signal is in the audible range for our ears to hear.

So far in the receiver we have been able to tune a signal with our VFO that was 6.9992 Mhz and turn it into an 800Hz tone on the receive end.

Gather the following components.

U3 NE612 mixer chip
C16 68pF
C17 47pF
C18 47pF
Y4 4Mhz xtal (the case of this one is not grounded)

Install on the board and check your work. Those of you with a scope can look at the output of U3 on pin 4 or 5. You should see an interesting mix of high frequencies (7.9992Mhz) and a low audio frequency (800 Hz).

For those without a scope, or if you just want to "hear" your receiver work, try this. Power off the rig. Get one of the 47uF caps left that has not yet been installed. Don't cut the leads! Tack solder the + lead to the C19 pad that is closest to U3. Now using clip leads connect one side of a speaker to the - lead of the capacitor and the other side of the speaker to ground. Make sure the room is really quiet. Power up the rig and hit the key. You should hear a very faint 800Hz tone coming out of the speaker. That is your superhet receiver at work! Cool, Huh?

Once you hear the tone, shut down the rig and remove the capacitor and put it back in the parts bag (you need it later).

There is not much to describe in this portion of the circuit. Just a standard implementation of an NE612 mixer.

Part 10

Up to now we have a functioning receiver. We have audio coming out of U3, but it needs filtering and lots of amplification. Let's build the first section. Gather the following components:

U4 NE5532 and socket
C19 .033uF
C20 .1uF
C21 .01uF

C22 150 pF
C23 150 pF
R2 10K
R3 10K
R4 510K
R7 510K
D3 1N4148
D4 1N4148

Install the components and check your work.

Turn on the rig and look at pin one of U4 with a scope when you key up. Hey, what's wrong here? I had much more signal at the output of U3 than I have here. What gives? Well, U4 is an amplifier, but it also has some limiting diodes in the feedback circuit, D3 and D4. These limit the output swing of the amplifier as to not saturate Q1, the mute FET (not installed).

Remember that up till now we are cheating with the receiver. We are using the transmit signal as a signal source to test our receiver. In the real world we would never have such a large signal on our antenna.

This section provides 30db gain and is differentially driven (like T2 at the output of U5) to take advantage of the greater signal voltage. It also acts as a low pass filter, rolling off frequencies that are over 1500 Hz.

R4 DC biases pin 3 of U4. C23 and C22 lower the gain of the stage as the frequency rises. Lets get a DC and AC analysis of this stage.

Part 10 Questions and Answers

Q. How can we calculate the DC voltages of this stage. Opamps should be easy, right?

A. Ok, it all starts with the +8v feeding in thru R4(510K)...that's the biasing reference voltage going into U4 pin 3. Since U4's bias current is so small, there'll be a very small D.C. voltage drop across R4, and pin 3 will end up with almost +8V too. Now at 0Hz (D.C. voltage), U4 has INCREDIBLE gain. That is, if there's ANY voltage difference between the two input pins (pin 2 and pin 3), the output D.C. voltage (at pin 1) will saturate: if pin 2 is ABOVE pin 3, the output will saturate down near zero volts... if pin 2 is BELOW pin 3, the output will saturate up near +12v.

Notice that there's a DC path from the output (pin 1) back to the "-" input (pin 2) through R7(510K). Once again, the D.C. current drawn into pin 2 through this resistor is tiny (insignificant). So to keep pin 2 = pin 3, the output will have to sit very close to +8V. So all three pins sit close to +8V. If you were to drag pin 3 down to +6V, the other two pins would follow. Isn't negative feedback wonderful?

Q. How do we calculate the gain of this stage. The manual says 30db. What is the voltage gain?

A. This is AC gain (DC gain is zero since C20 and C21 block DC voltages). AC gain is set mostly by the ratio of R7(510K) to R2(10K). This gives a gain of about 50. Actually, source impedance is a little higher than 10k, since we should add U3's output resistance of 1500 ohms (reducing gain a little). That's where the 30 dB comes from. But you can consider that gain is actually twice that, since BOTH output signals from U3 pin 4 and U3 pin 5 are used to drive U4 (each have the same AC amplitude). This is a differential amplifier configuration.

Q. Why is C21 smaller than C20?

A. To find the answer, you should look at the input impedance going into each input to U4 (this gets tricky): Input Z looking into R2 is 5K, not 10K as you might expect. Consider that when the input side of R2 is going UP, its output side is going DOWN by about the same amount. Remember, pin 2's voltage tracks pin 3's voltage...and pin 3 is driven from U3 with a signal of similar amplitude, but oppo-

site polarity. Input Z looking into R3 is very very high, since each end of R3 has the same AC voltage on it....except for the effect of C22(150pf) and R4(510K). R4 actually sets the upper limit on input Z here.

So C20 sees 1500 ohms on one side (from U3) and 5K on the other side (from R2). This works out to be a high-pass filter, letting through everything above about 350 Hz. But C21 can be a LOT smaller because impedance levels approach 510K. C21(0.01uF) just happens to be a handy value that is used elsewhere.

Q. What is the purpose of C19? Is this a terminator of some kind?

A. Remember that U3 is a double-balanced mixer. It gives about equal output at two frequencies:

L.O. + I.F. (4.0008 Mhz + 4.0Mhz = 8.0008 MHz)

L.O. - I.F. (4.0008 MHz - 4.0Mhz = 800 Hz)

We want the 800 Hz. stuff, but we DON'T want the 8MHz. stuff since it could cause U4 trouble (wasn't made to handle R.F.). So C19 "terminates" the R.F., leaving audio alone.

BTW: C19 and C20 and C21 were all ceramic capacitors in my kit. These are prone to generating "microphonics". And since they're ahead of a LOT of audio gain, they are the prime suspects for replacement if you have microphonic trouble. Check for microphonics: once audio is coming out the headphones, tap the board and listen for "boink" aftereffects. If you hear any, try tapping or scraping along these three caps to see if its worse. If you can run a fingernail along the face of these caps, and hear it in your headphones, then you might consider replacing them with mylar, or polyester or any plastic film-type caps of the same value. Should help reduce microphonics a lot.

Q. Will Q1 turn on without C24 installed? Is C24 just to add a delay in turn on?

A. Yes, you've got it completely right. R8 willdrag Q1's gate up to +8V, which turns the FET into a low-value resistor. Previously we mentioned about C24 "charging". It is R8 that does the charging, via the +8V available at U4a's output at pin 1. Time constant is 0.1seconds (1MEGohm x 0.1uf)

When the FET acutally turns ON depends on its pinch-off voltage (different for many kinds of FETs)...may not be exactly 0.1seconds. A neat experiment would be to short out R9 temporarily, to see how really LOUD the sidetone actually is (blow your ears off). And you'd hear the T/R thump too, although the sidetone may cover it up.

An Experiment

The best way to see what Q1 (the mute gate FET) actually does is to disable it and listen. Before proceeding, be aware that audio from headphones may end up being mighty loud...

Tack-solder a temporary jumper across R9(4.7MEG) on the bottom of the board. This will effectively "bypass" audio directly into the headphone amplifer, disabling the FET from its mute function. It won't hurt the FET. You should notice no difference while receiving; the FET is normally about 0.1Kohms while the key is up. 100 ohms is as good as a short-circuit compared with the 22K of R10.

But now when you hit the key, be prepared for a monster sidetone. The FET becomes a very high resistance once the key is down, but audio just shoots thru the short-circuit you've added instead. See how far back you have to turn the RF GAIN pot to cut the sidetone level. With the short-circuit removed, a little audio gets past the FET through R9, providing a rea-

sonably comfortable sidetone level. R8 and C24 are only in the circuit to provide some delay on key-up before the FET turns "on" and the receiver's audio comes back up. They have nothing to do with audio level, only timing. However R8 is required to drag Q1's gate up to the same voltage as its source. Vgs=0 volts is required for the FET to be "on".

Part 11: VFO Adjustment

Since we are close to getting our receiver finished, let's quickly adjust our VFO for operation in the amateur bands.

First, remove the temporary jumper that we installed before from Q6 to C36. This will remove our "cheat" and stop feeding xmit signal directly into the receiver.

You can use a frequency counter for this or a calibrated receiver. Using a frequency counter, connect the probe to the top of R29 or the cathode of D6. Turn on and key the rig, note the frequency. It should be around 7 Mhz. With a receiver, connect a clip lead to the top of R29 or the cathode of D6 as a small antenna. Turn on the rig and your receiver. Start at 7Mhz on your main rig and key the SW-40+. Tune the main rig up until you hear the transmitted tone. It helps to key it on and off so the carrier will be distinctive. Zero beat the receiver (tune it down until the receive tone gets so low you can't hear it, 0 Hz). The receiver is reading your transmit frequency.

Now, go to the manual page 16. Look up your frequency on the chart and find what capacitor that you will need for C7. Make sure you use the correct chart for Novice operation. Mine required a 68pF capacitor for C7. Install C7.

Next remove the temporary jumper at J2. Install the off board tuning pot (not included with kit) as per page 15 of the instructions. Now you can tune your xceiver. I built mine for the general+ segment. My freqency coverage is from 7.015 Mhz to 7.050 Mhz, perfect for me. Without C7 mine tunes from 7.111 to 7.149. Maybe I can install a switch on C7 to switch between General and Novice operation.

I don't remember if we covered this in the VFO section, but you can change the range of tuning by changing C8. Increasing C8 widens the tuning range and decreasing it decreases the tuning range. The manual states that the practical upper limit of C8 is 1000pF, and at the larger values capacitor and varicap thermal stability become crucial. Use NPO/C0G caps if you replace C8. Now that our VFO is set we are ready to finish the receiver and listen for some stations!

Part 12: Final Audio Amplifier

Ok folks, let's make this thing hear! We are going to complete the last section of the receiver that consists of the mute circuit and the final audio amp/bandpass filter.

During key down we want to reduce the signal to the final audio amp. Remember that a portion of our transmit signal is fed into the receiver to give us a side tone. We want to reduce the level to a comfortable listning level. On key down Q1 acts like an open circuit, so the only audio getting through is that through R9. This is why if you reduce the value of R9 that the side tone gets louder.

The second part of the circuit that consists of the other half of U4 is an amplifier / bandpass filter. It has approximatly 30db of gain and a center frequency near 800Hz. It also provides the power to directly drive headphones without needing and additional audio power amp. Those LM386s are noisy

The parts gathering should be getting easier as the contents of the parts bags dwindle.

R6 10 ohms

R8 1Meg
R9 4.7Meg
R10 22K
R11 510K
R12 1Meg
R13 1Meg
R14 10 ohms
D5 1N4148
Q1 MPF102
C24 .1uF
C25 820pF
C26 .0022uF
C27 47uF
C106 47uF
C107 .1uF

Install the above components. Be aware of the polarity of C27, C106 and Q1. Connect a set of low impedance headphones or a small speaker (good headphones are MUCH better) to J3 pins 1 and 2. Connect a temporary antenna to the side of C40 that does not connect to the 4 diodes D7-D10. The antenna should be long enough to pick up a strong signal, maybe 10 or more feet. Remember that we don't have the RF gain connected yet, so watch out for strong signals. This receiver is VERY sensitive. When signals are too high remember that they are clipped in the first audio stage by D3 and D4, so it shouldn't blow your ears out.

Try this in the evening when 40 meters is usually very active. Power up the rig. You should hear some hiss from the headphones. Tune around and listen for some CW signals. Cool, It hears! Glen Leinweber has done an outstanding circuit description of the audio sections. It is a detailed analysis of both the first and second stage audio circuits.

SW40+ Audio Circuits

A general-purpose dual operational amplifier (NE5532) is used to amplify audio to drive headphones. Most of the rig's gain occurs here. These audio circuits are detailed in many texts of op-amp applications. This note concentrates on aspects unique to the SW40+ application of these common circuits. Once again, circuitry is described in blocks: separated into power amp, mute gate and differential preamp.

Audio Output Amplifier

One of the two NE5532 op-amps is configured as a band-pass filter, and can drive headphones to a fairly high sound level. Its highest gain of 33 occurs at a frequency of 816 Hertz. Q is about three, giving a bandwidth of 270 Hertz. This bandwidth complements that of the crystal filter. It also reduces wideband hiss resulting from high audio gain that might be otherwise irksome. The components that determine frequency response and gain are **R10**(22K), **R13**(1MEG), **C25**(820pF) and **C26**(.0022uF).

Because the output at pin 7 is at a DC potential of +8v, audio must be coupled to headphones through a capacitor, **C27**(47uf). A "ballast" resistor is included, **R14**(10 ohms), to prevent strange reactive headphones from giving the op-amp a hard time.

This amplifier has a very low output driving impedance (less than one ohm), and will try to pump current into low-Z phones (or an inadvertent short-circuit) until internal current-limiting circuits kick in (maximum load current is 38 ma). Current limiting "clips" the positive-going and negative-going audio peaks, resulting in distorted, harsh audio.

The RF Gain control should be set so that audio level is below the current-limiting threshold. Set this way, current-limiting acts very nicely to clip the odd noise peak. You could call this a poor man's noise limiter.

With no feedback, this amplifier would have very high gain (over 10,000). The four feedback components mentioned above limit the maximum gain to 33. Don't think that the extra gain is "thrown

away",it is diverted to other purposes:

- distortion is reduced
- output is very low resistance, able to drive any load
- input impedance is linear and predictable
- gain & frequency response is determined by passive (external) components
- operation is independent of supply voltage, bias voltage and temperature

Reducing excess gain with feedback is a powerful way to improve desired amplifier characteristics - most of the amplifiers in this rig (RF as well as audio) use this technique. You can thank feedback for making sure the theoretical and measured frequency response shown above agree so closely.

The Mute Gate

Q1 is a switch. When it is "open", its impedance from source to drain is very high (many megohms). A small fraction of audio is allowed to leak from U4a to U4b through **R9**(4.7MEG) so that a "sidetone" can be heard. When it is "closed", the FET appears as a small resistance (roughly 100 ohms from drain to source), and audio is conducted through it from U4a(pin 7) to **R10**. You should be able to see that **R9** and FET switch are effectively in parallel for audio signals.

Why is this FET *not* an amplifier? Its drain and source are at the same DC voltage. This means that there is no standing DC bias current: the only current passing through this FET is due to AC signals. But doesn't the AC signal appear between source and gate, as in an amplifier? Consider the DC potential there as well... When the FET is "open" a full -8 volt DC potential on the gate overpowers any small AC signal on the source -nothing gets through. When the FET is "closed", the gate and source are connected together through **R8**(1MEG) keeping the gate at the same DC potential as the source. However, **C24**(0.1uf) bypasses the gate, so yes, audio signals on the source could affect the FET-switch operation. However, as long as AC signals are small (less than a few hundred millivolts), resistance between source and drain remains linear, and insignificantly small. You can't use a gate like this at high audio levels (at headphone levels, for instance) because of the requirement of small drain-to-source AC voltage.

The capacitor **C24**(0.1uf) is required to keep the FET gate in its open state after transmitting for a few hundred milliseconds to allow the receiver to "recover" and not pass an audible thump on key-release. C24's voltage is dumped very quickly through **D5** to cut off audio very quickly, preventing an audible thump on key-down. Sequencing transmit/receive switch over this way is critical for seamless audio with no clicks or thumps.

Audio Preamp, U4a

U4a is configured as a differential amplifier. A regular amplifier's input uses ground as its voltage reference. A differential amplifier has two inputs (neither one grounded): output is proportional to the difference between these two inputs. A good differential amplifier ignores any signals that are common to the two inputs, only amplifying differences. In this case, differential inputs are at **R2**(10K) and **R3**(10K).

This kind of amplifier is more complex, requiring more parts than a simple amplifier. Why did Dave choose it? The product detector **U3** provides two opposing-polarity outputs ideally suited for differential amplification. This means that audio amplitude is effectively twice as big. With so much audio gain following, audio hiss due to op-amp noise is significant. We should take advantage of all the signal available: a single-ended amplifier could only use one or the other output of **U3**, not both.

There is a more subtle reason for using the differential configuration that also involves noise. **U2** is a rather noisy voltage regulator - while its output is a constant +8v, AC variations (at 800Hz) are significant. So **U3's** supply (pin 8) contains audio noise that propagates through internal 1500 ohm resistors to its outputs at pin 4 and pin 5. Since this noise is common to both outputs, a differential amplifier will ignore it as a common-mode signal. Remember, **U4a** will only amplify *differences between* the two outputs of **U3**.

Gain of **U4a** is mostly set by the ratio of two resistors (**R7** / **R2**). That's 510K/ 10K or a gain of 51. Differential gain is twice this value (102). **U4a** is also a simple low-pass filter, with **R7**(510K) and **C23**(150pF) providing a roll-off frequency of 2080 Hertz. Diodes **D3** and **D4** clip large amplitude signals, keeping output signals small enough that the mute gate can handle them. These diodes also help keep U4a stably biased, even for very large input signals.

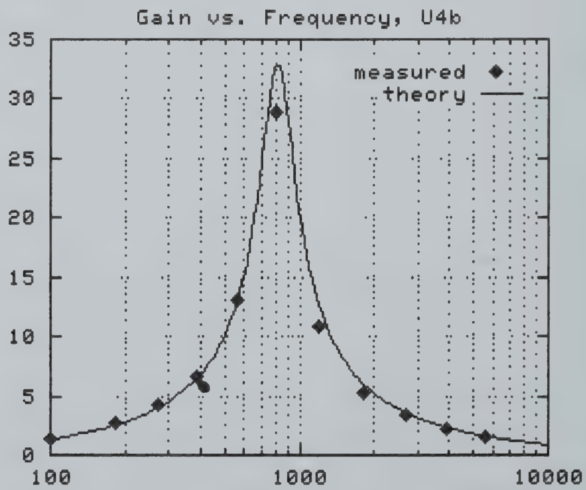
Audio from the product detector is fed into the differential amplifier through coupling capacitors **C20**(0.1uF) and **C21**(0.01uF). **C20** and **R2** act as a high-pass filter with cutoff frequency of about 350 Hz. **C21's** effect on frequency response is insignificant since U4a's input resistance on this leg is so high.

Glen has done some excellent work here. Please read through it and be prepared, bring your thinking cap. All we have left is the power amp and output filter.

Part 12 Questions and Answers

Q. I rewired my phone jack so that the headphones would be in series rather than parallel as suggested in "QRP Power", page 3-22. The Op amp seems to drive my phones louder before clipping. It really seems to work for me. I gotta wonder why it wasn't designed this way, is there a draw back that I'm not aware of?

A. I agree, that's the way to wire headphones, especially when driven from a feeble driver like an op-amp. But there IS a catch. You must be careful not to ground the headphone jack. The shell of the jack is the common-point between the two headphone elements. When you wire each



element in series, then one "hot" element-end ends up going to the ground-plane on the printed circuit board. The headphone jack shell is now at a half-voltage point.

If you try to wire the jack in a series-connected way, and find you only get audio out of one side of your phones, the cause could be a grounded shell. Insulating the jack from ground should solve the problem. Or using a plastic (insulating) case to mount everything solves the problem too.

Audio Experiments

Here are some experiments to try on your SW40+ receiver to help determine noise sources in the rig. Just how much audio noise is internally generated, and how much is due to RF noise coming in the antenna?

Experiment #1. Pull out U3(SA612) with the power off. With the rig powered back up, you'll hear low-level audio hiss. Take a mental note of its level, or measure its amplitude on your scope. This hiss is entirely due to noise being generated within the high-gain audio op-amp stages of U4. It isn't very loud. Power off, replace U3 (careful to place pin 1 correctly) and power up. Compare the "hiss" level: significantly louder. Even though the bulk of this rig's gain is in audio stages, most noise you hear is RF noise. Good.

You could do this experiment as well by carefully jumpering pin 4 to pin 5 of U3. I did this with power on, holding a short insulated wire. Be careful not to short adjacent to pins.

Experiment 2: Take an electrolytic capacitor (I had a 47uF/16V handy) and bypass U3 pin 4 to ground. GET THE CAP'S POLARITY RIGHT! Try bypassing pin 5 to ground too. Since half the audio signal comes from pin 4, and the other half from pin 5, you'd expect noise level to drop by 3dB. But I hear noise level go up! Careful listening revealed that noise BANDWIDTH went up too. What's going on?

This is tricky (hope I've got it right). If you listen carefully, you'll hear the narrowband noise (400Hz wide) drop in level, as expected. But wider bandwidth noise is introduced that brings the apparent overall noise level up. Where did this noise come from? I believe that it comes from U2, the 78L08 regulator. It sends wideband noise into U3's power supply pin 8. This noise shoots thru a 1500-ohm resistor (inside the chip) into the audio amp. When you DON'T short one of U3's outputs to ground, this noise appears IDENTICALLY at BOTH outputs of U3. Since U4a is a differential amplifier, it rejects U2's noise as a common-mode signal. U4a only amplifies difference-signals between pin 4 and pin 5 of U3.

Experiment #3. Listen to the noise level while you unplug your antenna. Noise should drop. This is a quick-n-dirty test to determine if a receiver has enough "sensitivity". With no antenna, you're listening to noise generated entirely within the receiver.

Here are some other "safe" points in the receiver that you could try shorting to ground with a short insulated jumper:
-link winding of T1 (J1 pin 3)
-either end of RFC1
-Junction of Y3(4MHz) and C15(150pF)
Glen VE3DNL leinwebe@mcmaster.ca

If you are having trouble peaking T1 for max audio out:
My rig would not peak. The slug in T1 was all the way out and I did not reach a "peak" in the signal. I soldered a 22pf capacitor in parallel with C1, which took the total capacitance to 69pf. I got a nice peak at about mid range of the slug.

If you are having trouble peaking and the slug is all the way out, try adding capacitance. If the slug is all the way in, try reducing C1.

Part 13:Final RF Amplifier and Filter

This is the last construction section The last part of the kit is the final RF amp and output filter.

FROM THIS POINT ON MAKE SURE YOU HAVE A SUITABLE ANTENNA OR DUMMY LOAD CONNECTED TO THE ANT PAD ON THE CIRCUIT BOARD. Transmitting without a resonant antenna or dummy load will cause high SWR and can damage the final transistor. This is why we saved this section for last. An outstanding description of this amplifier and it's function was done by Glen, VE3DNL.

SW40+ Final Power Amplifier (Q6)

QRP power amps are not complex: you'll see from this note that modeling the transistor as a SPST switch gives accurate results. A PSpice switch model is compared with a full-blown PSpice transistor model, and then compared with oscilloscope waveforms taken from a working SW40+ .

Q6 substituted with a switch?

Yes, let's try this simplifying model first. Q6's collector/emitter is substituted with a SPST switch. PSPICE controls the on/off state of the switch from a control voltage. So our simple switch actually looks like a four-terminal device: two terminals are the actual switch, while two terminals accept the control voltage. In our case, the control voltage **V1** is a 7MHz. square wave so that the switch **S1** is *on* for a time period of 71.4 ns. and *off* for another 71.4ns. **V2** is the +12v D.C. power source. **Rantenna** represents a 50-ohm dummy load. All other components show their SW40+ schematic designations. Remember, **S1** substitutes for the collector-to-emitter connections of **Q6**.

The PSpice waveforms show voltages at the switch [**V(L4:2)**] and at the 50-ohm dummy load [**V(C36:1)**]. It should be clear that when the switch is on (short-circuit), its voltage is clamped to zero. When off (open-circuit) switch current is zero, and voltage at the switch terminal can float wherever it wishes. The floating voltage is constrained by choke **L2** and the following PI network components to look like a half-sinewave, swinging up over the +12v supply voltage, before the switch turns *on* again. Note that the average voltage at the switch must be equal to the supply voltage of 12v. The inductor **L2** requires this to be true. When the switch is on, it temporarily drags **L2** (and **C36**) down to ground. Then the switch opens: voltage must soar above the supply in order to keep **L2**'s average voltage at 12v. That's why peak voltage rises to about 34v. The combination of **C36** and the 50-ohm dummy load resistor must result in average voltage at the load of zero. So the dummy load voltage swings about zero volts: rising to +17v and dipping to -17v . The five-section PI filter consisting of **C37**, **L3**, **C38**, **L4** and **C39** accepts only the 7MHz energy and rejects most of the higher harmonics. The result is a clean sinewave at the dummy load shown as **V(C36:1)**.

PSpice transistor modeling

How accurate could the simple SPST switch model be? Let's do a more complete SPICE model that includes not only a transistor for **Q6**, but a proper driving circuit too. The final amp is actually a 2N4401 transistorscaled up in size. Q2's collector drives the primary of a transformer. Coupling between L3 and L4 is tight (99%) as a ferrite toroid should be. An attempt was made to simulated lead inductance at Q1 with 5 nh inductors (L6 & L8). This model is very close to the SW40+ schematic, however, the parts numbering is different. Now let's compare the three cases: the collector voltage of the final amp from the PSpice model above, the actual waveform as measured on an oscilloscope, and the simple switch model. The switch model (bottom) is very similar

in amplitude and shape to the more complex transistor model (top). The oscilloscope trace (middle) has slightly lower amplitude, is a bit more jagged, but maintains the same shape. Note that peak voltage rises up to nearly 30v. The zener diode **D12** would clip anything more than 33v, protecting **Q6** from overvoltage. Should you decide to raise the supply voltage, **D12** should be swapped for one of higher voltage. **D12** should never conduct during normal operation: normally, it only contributes a small capacitance. Note that circuit operation depends almost entirely on passive component values, not on transistor characteristics. The PI filter is nearly symmetrical so that (at 7MHz.) the transistor "switch" sees a 50-ohm non-reactive load. If the transistor switches efficiently, and component losses are ignored, then we'd have a 12-volt peak square wave applied to the filter. With harmonics rejected, that works out to 1.44w RMS out the antenna.

Is Q6 a class C amplifier? Here are some "classic" definitions of class C operation:

· With no input drive, no collector (plate) current flows

· Collector (plate) current flows for less than one-half cycle

· Collector (plate) voltage shouldn't saturate

The SW40+ final amp only satisfies the first point. Collector current flows for close to half-a-cycle (perhaps a little more). Collector voltage saturates down to zero volts. Some folks use the term saturation in slightly different context: if input power is increased, the amplifier is said to be saturated when power output no longer increases. At this point, the amplifier has long before hit ground on the negative-going swing (voltage saturation). When cranked up, the SW40+ still has a little more to offer before power saturation, but not much. I'd hesitate to use the "class A/B/C/D" criteria of operation. Dave has attempted to make the final amp run as efficiently as possible. You can get an idea of how little power is wasted by feeling how cool **Q6** is, and by the fact that no heatsink is required. Half-cycle conduction, nearly full saturation, and operating into a high load impedance all contribute to high efficiency. Following the "class-C rules" above would result in wasted power.

Q6 base drive

What's it take to drive **Q6**'s base? It is very low impedance. Voltage here never gets very large because **Q6**'s base clamps to about +1v, and diode **D6** clamps to -1v. You might think that **R29**(51 ohms) establishes the base drive impedance, but it is nearly ten times larger than actual impedance of about six ohms. The one-turn link winding on **T4** provides a low driving impedance for Q6's base.

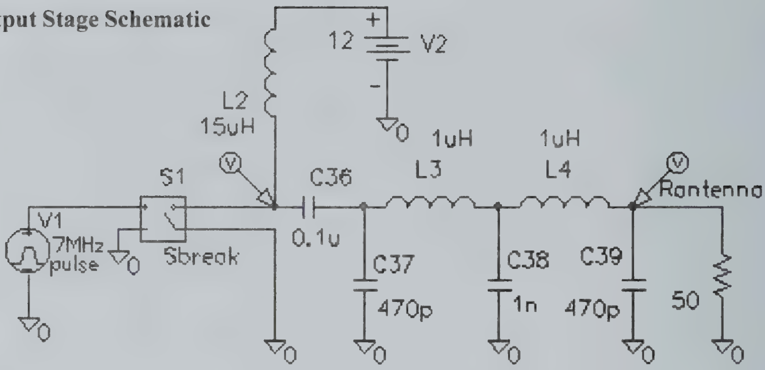
Before I read that page, I would have described Q6 as a class C amplifier. Glen shows that the lines are a little blurry when defining an amplifier in a class. Glen modeled Q6 as a simple switch, on for one half cycle of the 7 MHz and off for the other. As you will see the model comes really close to what was actually measured.

I'm not sure about this, but it seems to me that with Q6 either full on or full off that it's power dissipation must be fairly low. This is also reflected in the fact that Q6 has no heat sink. Operating my SW-40+ in a normal QSO, Q6 barely got warm to the touch.

As for the output filters, they are described as a "5 element Chebyshev" filter. Back in March Chuck posted and excellent tech. description that I am going to include here. The following was done by Chuck Adams with parts by Paul Harden.

A March 26th posting by Paul Harden on low pass filters prompted this posting. C36 is a 0.1uF and is a simple coupling cap, that is, blocks DC from the final transistor (PA) from the filter components.

Output Stage Schematic



L3,L4 and caps C37, C38 and C39 for the output filter, which is basically TWO low pass filters glued together (hopefully with solder!). The values are:
L3, L4 = 1uH (16 turns on a T37-2 toroid)
C37,C39 = 470pF
and C38 = 1000pF

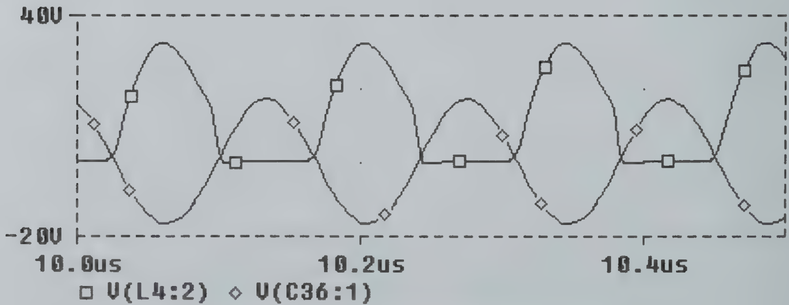
At the desired frequency of operation, if you will make the impedance's XC (37) = XC (39) = 50ohms, XL(L3) = XL(L4) = 50 ohms, and XC (38) =25 ohms to get 50 ohms to 50 ohms impedance match. You can see why C38 has twice the capacitance of C37 or C39.

Here is a table for L and C values that will give you 50 ohm reactance at each of the most popular QRP frequencies.

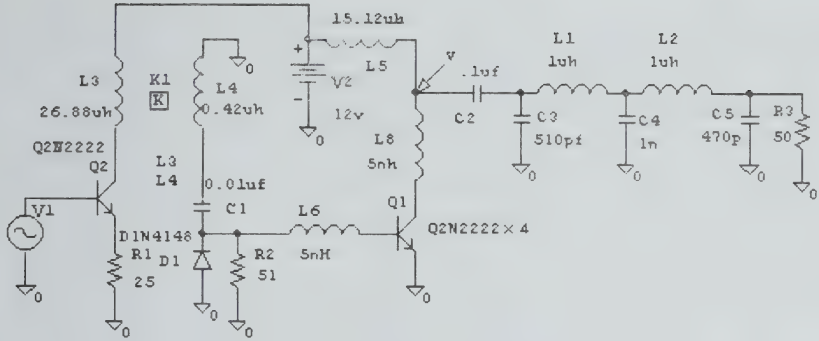
Freq [MHz]	**L [uH]**	**C [pF]**
3.560	2.23	894.1
3.710	2.14	857.9
3.579	2.22	889.3
7.040	1.13	452.1
7.110	1.11	447.7
10.106	0.78	315.0
10.116	0.78	314.6
14.060	0.56	226.4
18.080	0.44	176.0
21.060	0.37	151.1
24.910	0.31	127.8
28.060	0.28	113.4

*C = capacitance for end caps. Double value for center cap.

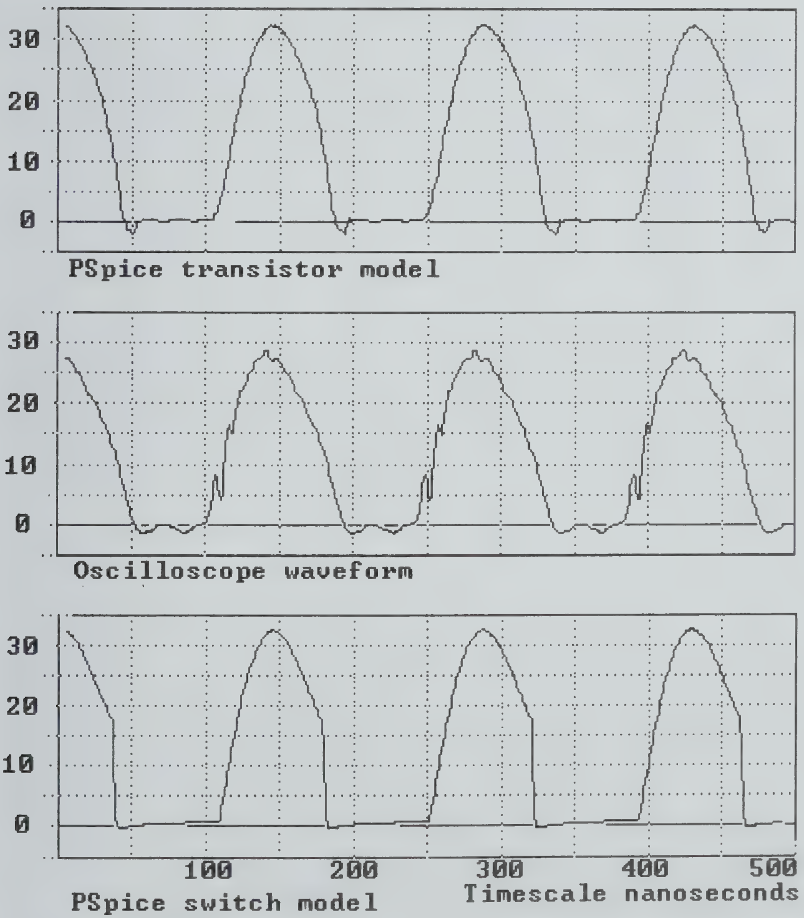
Now the C values are not exactly off the shelf standard values, so you try for the closest you can get. Let's use the 40 meter frequency of 7.040MHz for an example. Dave chose 470pF for C37 and C39 and 1000pF or 0.001uF for C38 and L3=L4=1.02uH. This because 390pF is close on the low end and 470pF is closer to the desired value of 452pF and 0.001uF



PSpice Switch Simulation Waveforms



Q5 - Q6 Schematic



Comparison of PSpice Transistor Model, Scope Waveform & PSpice Switch Model

is closer than 820pF for C38. See Paul Harden's book for the standard capacitor values. These values are also the same values recommended by Wes Hayward in the SSD book. (Solid State Design for the Radio Amateur an ARRL book)

For the inductors, we get for the two popular toroid sizes used in QRP work and the two most popular cores:

	T37-2	T37-6	T50-2	T50-6
# turns	[uH]	[uH]	[uH]	[uH]
1	0.00	0.00	0.00	0.00
2	0.02	0.01	0.02	0.02
3	0.04	0.03	0.04	0.04
4	0.06	0.05	0.08	0.06
5	0.10	0.08	0.12	0.10
6	0.14	0.11	0.18	0.14
7	0.20	0.15	0.24	0.20
8	0.26	0.19	0.31	0.26
9	0.32	0.24	0.40	0.32
10	0.40	0.30	0.49	0.40
11	0.48	0.36	0.59	0.48
12	0.58	0.43	0.71	0.58
13	0.68	0.51	0.83	0.68
14	0.78	0.59	0.96	0.78
15	0.90	0.67	1.10	0.90
16	1.02	0.77	1.25	1.02
17	1.16	0.87	1.42	1.16
18	1.30	0.97	1.59	1.30
19	1.44	1.08	1.77	1.44
20	1.60	1.20	1.96	1.60
21	1.76	1.32	2.16	1.76
22	1.94	1.45	2.37	1.94
23	2.12	1.59	2.59	2.12
24	2.30	1.73	2.82	2.30
25	2.50	1.88	3.06	2.50
26	2.70	2.03	3.31	2.70
27	2.92	2.19	3.57	2.92
28	3.14	2.35	3.84	3.14
29	3.36	2.52	4.12	3.36
30	3.60	2.70	4.41	3.60

Dave went for the 16T on a T37-2 for a value of 1.02uH. He uses 1.00uH in print and that is close enough due to variations from core to core on the A(L) value.

I did a quick SPICE simulation and here is my first recommendation for a SW-40+ mod. You knew the mods were coming, didn't you????? Add one more turn to L3 and L4, thus 17T, and this will increase the second order attenuation by an additional 4dB without significant penalty at the fundamental. And I even modeled in some additional distributed capacitance due to packed turns and it seems to be for the good in this case. I get 34.85dB down at 14.080MHz for 17T vs 30.79dB down for 16T. Now as soon as some of the people get theirs built and can get them to a lab with a high-dollar spectrum analyzer they might take the time to try it both ways and tell me if their is any difference. Probably not enough to worry about, but it would be interesting to look at theory vs real data. The SW-40+ etc. all work great as designed and built from the instructions, so this recommendation may be ignored. Due to other factors yet to be studied (see below) the input impedance of the final PA may influence this effect.

Now those that are going to move their SW-40+ rigs to the Novice frequencies on 40M will start to ask questions on what mods do they need to do? The answer is none, as the filter is not critical for the small change from the low end of 40M to the higher end.

In the latest issue of QRP ARCI Quarterly you will see a graph of attenuation curves from a bunch of filter values that I took from all the W1FB books and Dave's filter. NN1G's is the solid curve. The only one that beat it was the theoretical maximum. Good job Dave.

Those of you that went to the trouble of getting SPICE up and running on the computer can check out what happens when you vary some of the values.

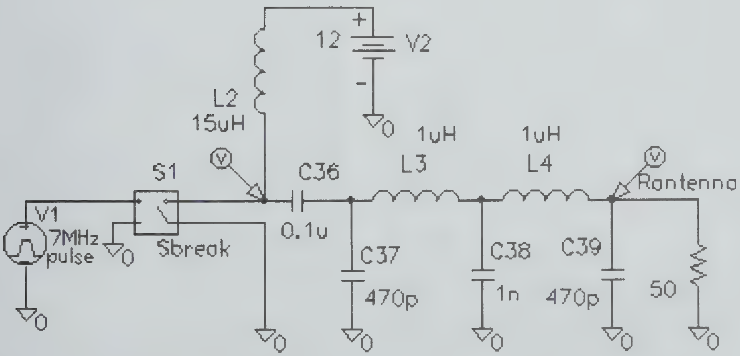
One more thing to look at is relative to postings that I see where people 'squeeze' the windings together for these type filters and see power output increase

from a rig into a dummy load. Personally, in my opinion, this is a dangerous practice. Into a dummy load, the second and third harmonics (which may increase) contribute to the forward energy detected by the SWR bridge and or power meter. BUT, into a real antenna you may see the reflected power increase at the same time, thus showing that the effect is not what you really want. Again, this is something else to experiment on. Remember, mileage may vary. You have to take lots of data and that's what makes it all interesting and challenging. For those that like to read:
"Ferromagnetic-Core Design & Application Handbook" by Doug DeMaw, W1FB, published by MFJ Publishing Company, Inc. Starkville, MS 39759, MFJ-3506, $19.95
"Simplified Practical Filter Design" by Irving Gottlieb, published by TAB Books, Blue Ridge Summit, PA 17294-0850, ISBN 0-8306-3355-3, $16.95. and of course the ARRL HB and Paul Harden's book.

Now back to building. Wind L2, L3 and L4 torroids as per the instructions in the manual. Remove the temporary jumper from J1 and install the proper 5K RF gain pot. Gather and install the following components:
L2 as per instructions
L3 as per instructions
L4 as per instructions
C36 .1uF
C37 470pF
C38 .001uF
C39 470pF
C113 .1uF
D12 33 V zener diode
Q6 2SC2078

You will have some capacitors left over. That's ok. They were included by Dave for the VFO calibration. Check your work. Be aware of the polarity of D12 and Q6. Don't install them backwards.

Connect a 50 ohm dummy load to the ANT connection and ground. I just soldered some wire to an SO-239 connector and soldered the other ends into the ANT and ground holes. The antenna connection should go to the center conductor and the ground should go to the shield. I then connected the SO-239 connector to the dummy load in the shack. If you have a QRP watt meter you can put it between the rig and the dummy load to check your RF power out. If you don't have a watt meter, con-



Output Stage Schematic

nect your RF probe or oscilloscope across the dummy load to view the output.

Turn it on. You should hear static from the headphones. Turn your RF gain to maximum. Key the rig. You should see your watt meter move to about 2 watts and hear a nice side-tone. O-scope users should see about 30 volts peak to peak of RF. You can adjust the R24 potentiometer to vary the power out. I turned mine up all the way and the waveform got distorted. I backed it off until it looked clean and then turned it down some more. Now I'm measuring 30 volts p-p which is about 2.25 watts out. Remember that peaking the power into a dummy load is not the best way to get the most power. You may be increasing one of the harmonics and the actual useable power may go down. Also your rig may not be in FCC spec for spurious emissions. If I had a spectrum analyzer I'd like to see how the harmonic output changes with R24 position. Well, that's it. If your power out looks good, put it on your 40 meter antenna and make a contact. My first two contacts were Texas, then Cuba. Not bad for 2 watts.

I put my board in the case from a computer A/B switch box. I mounted the RF gain, tune pot, phones and key jacks in the front. On the back I put the antenna SO-239 and the DC power jack. Right now I don't have an on/off switch but I think I may add one to the RF gain pot. I used a 10 turn Spectrol linear pot for tuning. I also got one of those small 10 turn indicators. I thought it would be easy to calibrate, but it seems that the varactor response is not linear. The frequency difference between major divisions is 2kHz on the ends and 5kHz in the middle. So all I did was make a little chart on the front of the rig that had the operating frequency for every one of the 10 major divisions. Enough to get me close. I can tune between 7.015 and 7.050 Mhz. I think the first addition to this rig will be a Small Wonder Labs "Freq mite" for a CW frequency indicator. This radio is a joy to use. It is small and uses very little power, yet the receiver is very sensitive.

Part 13 Questions and Answers

Q. I'm not sure about this, but it seems to me that with Q6 either full on or full off that it's power dissipation must be fairly low. This is also reflected in the fact that Q6 has no heat sink. Operating my SW-40+ in a normal QSO, Q6 barely got warm to the touch.

A. Absolutely right. Only reason Q6 gets warm is that it doesn't switch "on" instantly and "off" instantly. Dave has designed the final for very high efficiency: Driving with 50% duty-cycle helps. Saturating Q6 helps too. Keeping the load (that Q6 sees) as large as possible helps too. Dave gets maximum power out using minimum DC current. This is very important for QRP rigs that might run from batteries. A Class C amp that followed all the "class C" rules wouldn't do as well.

Final Questions, Answers and some Troubleshooting Tips

I traced the low sensitivity problem I was having with the SW40+ to Q1. Good audio on the source and low audio on the drain. Had some MPF102's on hand and replaced same. Problem solved! Now, no matter where I put the probe around the bench, I have to turn my signal generator off to not hear it. The keying thump also went away. Guess that was the clue all along.

I tacked an 8.2pf NPO (had one) to the board, across C1 and can now tune T1 for a peak.(K8IQY changed C1 to 68pf to get a peak and Glen Leinweber mentioned that he gets T1 to peak with core turned near the end.)

Q. I am building a SW-40+ (with the QRP-L Elmer 101 project). I got the transmitter working with your help. Thanks! It

puts out a nice, clean 2+ watts. About 30 V peak-to-peak. Very nice signal.

Then I couldn't wait any longer, and went ahead and finished the kit! I have a problem. Now when I transmit, I get very little power out of the transmitter. (I haven't even tried to see if the receiver works yet.) I only get about 150mv peak-to-peak out when keying the transmitter. I am trying to understand the TR switch circuitry, and the 4 diodes (D7 - D10). It almost looks like the diodes will limit the voltage. Then I calculated the impedance of C40 to be 500 ohms at 7MHz, so I guess if I can get enough current flow through RFC3, the combined IR drop across C40 and the voltage drops from the two diodes should still be OK.

It isn't. Do you have any ideas what could be wrong? I did install the potentiometer at J1 and peaking T1 does affect the transmitter output very slightly. RFC3 is not "open". I have not installed the tuning potentiometer at J2 yet. but still have the jumper from pin 2 to 3.

A. C40 and RFC3 form a series-resonant tuned circuit, which resonates around 7MHz. Its low-Q, so doesn't need to be trimmed. So even though you calculate 500ohms for C40, the actual impedance between the antenna PI filter and the RX input (T1) is very low. With a properly working transmitter, D7-D10 will conduct on the transmitted peaks, limiting the >30v to about 2.6v p-p. When the diodes conduct (while transmitting), their impedance is VERY low. This destroys the Q of C40/RFC3, and you can't think of these two components as a series- resonant circuit anymore. Now RFC3 acts as a choke of 440 ohms impedance going into the receiver.

What does the transmitter see? It mostly sees a 47pf capacitor going to ground - the diodes provide a ground-path for a large portion of the 7MHz cycle. You could ground the diode-end of C40 and the transmitter wouldn't notice (but then the receiver wouldn't get any signal).

In troubleshooting your transmitter, I'd look elsewhere for the trouble, not at this part of the TR circuit. I'd be looking around Q6... at D12, D6. I'd want to know if Q6 is getting a good base-drive signal. That'd tell me if the trouble is before the final. If there IS a good 7MHz signal at the base, I'd be taking a close look around Q6, D12, L2, and the PI-network to the antenna. Hope you've got a dummy load on, while testing.- Glen

Q. Thanks for the help! Here is the latest in the saga. First of all, yes, I do have a dummy load connected. I checked the voltage at the base of Q6 and get a nice, solid 2 V p-p at 7MHz. The voltage at the collector of Q6 remains a meager 150 mv p-p. This pointed me to suspect D12. I lifted one end of this diode so I could check it with my Ohm meter. I get about ~74 ohms(??) in one direction and "infinite" in the other direction. I think this indicates that it is OK, but it may still be the culprit. I don't have an easy way to test the reverse "zenner" action that this diode is supposed to provide. I don't have another 33V zenner handy, but will get one if you think that is still a prime suspect. I haven't tried transmitting with D12 lifted. I don't think it would hurt anything, and didn't dare!

I tried grounding the diode end of C40, and the transmitter output still remains the same. I have checked all the DC voltages that are indicated on Dave's schematic, and all are good. L2 appears to be good; the 150mv p-p is found on both sides of L2.

The thing that is so confusing to me is that the transmitter WAS WORKING FINE until I added the receiver and TR circuitry. This makes me suspect a component that was ADDED. Not necessarily, I know. Is it correct that I should see about

30v p-p from the collector of Q6 to ground? (Same as the output of the PI network.)

A. Since you've got D12 lifted, try the transmitter. Just be sure your dummy load is connected: the purpose of D12 is to protect the final amp from rouge loads. 74 ohms seems a little low for measuring the impedance of the zener. But some ohmmeters display strange stuff when measuring diodes, depending on their current source, and voltage scaling. Since it DOES measure infinite resistance the other way, I'm with you - I think it's probably OK.

Looks like you've got good base drive on the final. Its looking more and more to me like the final amp (Q6) is cooked somehow. Before pulling it out, do a really careful visual inspection on top of the board, and on the bottom of the board around Q6, L2, D12, C36, C37, C40 and L3. Look for open circuits and shorts. I'd do this check under a really bright light and a magnifying glass. I've found more than a few faults this way: things like hair-line cracks in traces, or lifted pads or filamentary shorts between traces. Measure the DC voltage on the heat-sink tab of Q6: should be +12V DC. Measure the base drive right at the base lead of Q6: you may have measured it elsewhere, and somehow its not getting into Q6. Also, make sure the emitter lead is well-grounded too - its not a check that Dave has asked for, but the final amp sure won't work if Q6's emitter isn't at ground.

Yes, the peak-to-peak voltage at Q6's collector should be in the range of 20-30 volts, unless you've got the drive pot turned way down.

The next step after doing all these checks is to pull Q6, and test it with your ohmmeter. De-soldering a three-legged transistor like this is tricky: the printed circuit pads take a beating. Its an area where currents are high, so you don't want to beat-up these pads too much. Sometimes its better to cut Q6 out rather than de-soldering it. Then you can clear the three holes one-by-one. Good luck, and keep in touch.-Glen

Q. YES! It works! The bad guy was Q6. I changed it, and now the transmitter is putting out a nice, clean 2+ watts again. Getting almost 30V p-p on output! Now on to testing the receiver. Glen, thanks so much for your time and your patience. In retrospect, debugging a problem like this has helped me understand the circuit more, and that is why I am building this. I really was confused though, and I really appreciate your depth of knowledge and willingness to help.

Q. My sw-40+ tunes from 7.0010 to about 7.036. When I tune in the low part(7.0010-7.0017), I hear a high frequency whine. It is now very loud, simply a bit over the background noise. What is causing it? Where does it comes from?

A. If the whine is tunable, and is at its lowest pitch at the low end of.the band. It's a 'birdy', or spurious mixing product. The product detector is sensitive not only to its 4 Mhz LO, but to oddharmonics of that LO signal as well. As you tune to 7.000, the 4th harmonic of the (3.000 mhz) LO beats with that spurious 12 Mhz product detector response to produce an audio output. If the pitch of the whine isn't tunable, it may be an intermod (IMD) product and can be eliminated by backing down the 'gain' control a bit. If that doesn't take care of it, it's an IF response, and can be minimized by 1) grounding the crystal cases ans 2) buttoning up the board in an enclosure. The diagnostic for this involves touching the crystal cases with a fingertip to see if this makes the whine louder.

Q. I will reverse the tuning pot wires... Right now, it is set in a reverse way (increasing freq. by going counterclockwise). I will probably do the same for the gain. Louder counterclockwise...Did Dave re-

verse things on his drawing coming with his enclosure? Or did I simply reversed the pots? (solder posts up instead of down...)

A. My original hookup illustration was incorrect- it's been fixed. (That presumes that 'clockwise' should correspond to'increasing frequency', of course.)

Q. I had to play around with L1 to get the desired range. Any suggestions on how to "cast it in concrete" to make sure it doesn't drift due to shock and vibration? What material do you people use, and how do you go about pouring/installing it without getting goop all over the rest of the components? Second, on receive I've got considerable hum in the background even with the volume control all the way down. Any way to get rid of it? Third, when I turn off the power, the rig gives a loud squeal through the headphones — loud enough to be quite uncomfortable if you're wearing the phones. Again, any way to get rid of this electronic flatulence?

A. Use clear finger nail polish on the toroid(s). Cheap and it works great. On the hum. You did not say what you were using for P/S. Battery? Otherwise check the DC output from other P/S's. P/S = Power Supply. The "I'm turning off patented NN1G feature" of the SWL-XX and SW-XX+ series is due to the audio section becoming an oscillator as the voltages and currents wind down.

Q. In Mike's final Elmer101 installment, he warns against keying the rig with a poor SWR at the antenna connector, indicating it could wipe out the final transistor. I've seen this same warning many times before. Could somebody please explain to me the mechanics of what's going on? Why should the transistor care what kind of SWR is out there? What happens in terms of emitter/collector voltages and currents that destroys the device?

A. Going to take a crack at answering. Haven't smoked a rig this way myself, so am going to apply some theory. When you've got a 50ohm resistive load at the antenna jack, then Q6 sees close to 50 ohm NON-REACTIVE as well. This is because the PI-filter consisting of C37, L3, C38, L4, C39 transforms the 50-ohm antenna into a 50-ohm impedance at its other end (at the transistor). This is true only at one frequency...7MHz. Everything runs great this way.

Now suppose you pull the antenna off. The PI network was designed for 50 ohm load. Now the load is infinite. Q6 sees a whack of reactance now, when it used to see about 50 ohms. Or suppose you short the antenna out. You've STILL got a whack of reactance at Q6's collector, although it's value will be different than the case where load is infinite. I modeled Q6 as a SPST switch (to ground) that opens and closes at a 7 MHz. rate. What's the switch see on its collector? A whack of reactance...perhaps inductive reactance from L2. This would look like a flyback switch (like a car's spark-plug coil and distributor):

During the time that Q6 is ON, current climbs (linearly) in L2, as current passes thru L2, thru Q6 to ground. Then Q6 turns OFF. L2's current goes to zero, and the voltage across L2 swings w-a-y above the supply voltage. Bad news for Q6, which can't take a big over- voltage on its collector. D12 saves the day by clipping voltage excursions on Q6's collector to +33 volts.

I once saw +160 volt spikes on the collector of a transistor operating in a circuit like this (with a 12v DC supply). The circuit became unstable because of the high (and reactive) load impedance, and turned into a low-frequency oscillator with these huge, sharp spikes. Luckily, the device survived.

When you've got a proper antenna load, the load resistance "damps" the wild

voltage swing, so that it doesn't even get high enough to cause D12 to conduct. In this case SWR is low, because you've got a load close to 50 ohms.

Well, that's it folks. Thanks for sticking with me. I can say personally that I've learned a TON of stuff about this radio and QRP rigs in general.

Blue Printing the SW40+ and the SW30+

by Gary Surrency, AB7MY
gsurrency@juno.com

None of thesechanges by themselves add that much to the output level, but when they are all added together the increased output level is significant. I hope this info is useful to those who are seeking the full QRP "gallon". If less output is desired, the drive level control R24 may simply be usedto reduce the output level and current consumption, as when operatingportable and using battery power. The stability and output spectrumremains clean at any power level up to maximum, and the PA transistorbarely warms up using the smallest heat sink that fits the TO-220 2SC2078package.

I changed D13 from a 1N4001 to a 1N5819 schottky to reduce power supply drop. I changed Q3 from a 2N3906 to a 2N4403 for less Vce drop. Keying looks better on the scope too, and it is the complement to the 2N4401, with higher hfe and Ic rating. I used a MPS2222a from R/S for Q5. A very nice, low input pf, high gain version of the 2N2222A. Many of my QRP rigs have benefited from using this excellent device that is widely available from Radio Shack and consistently a good part.

I added one turn of wire to L4 to alter the low-pass filter slightly.ChuckAdams mentioned something about this in a SPICE posting. Adding a turn toL3 didn't help the output or spectral purity. L3's 16 turns are evenly distributed around the entire core just like L4's 17 turns.

Used original 2SC2078 for the PA, as the MRF476's I tried produced no increased output. Used small heat sink on the PA, as the increased output did result inmore heat to be dissipated. A very small one will fit without modifyingthe PA's placement or orientation, as Dave mentions in the manual.

Used a NTE5087A 43V zener for D12, since the stock 33V zener shunts too much power when running 4-5 watts.

Used a 105 pF S/M cap at C8 to get 7.000 to 7.050 Mhz frequency coverage. C7is 10pF. I also used a quality .001 uF 100 V disc for C38 in place of the small monolythic that was supplied in the kit. I checked all the caps in the low-pass filter with a AADE LCIIB L/C meter to insure they were withinspec. There were no other changes, except I picked the highest gain 2N4401 that was in thekit and used it for Q4 following the TX bandpass filter for maximum output.

A lot of tweaking of the bandpass filters will achieve almost 6 watts into a 50 ohm dummy load as indicated on a OHR WM-1 wattmeter when powered from 13.8 volts DC. I notice some heating of the collector choke, L2, so there is probably some room for improvement there also. Not sure if it is core heating or merely heating of the small diameter enamelled wire. Still have toinvestigate this. Obviously some power is being wasted if heat is present. The rig has a very clean output signal and is less sensitive to antenna load changes than my NC40A is. Smooth QSK from the lil bugger too.

For those of you who want more output than the standard 2-3 watts the SW30+ is capable of stock as it comes from Dave, I have succeeded in eeking out an additional couple of watts. Not only that, the

rig actually became more stable with the mods that I came up with.

Now for a little history:

After I built the kit, I could get maybe 3-4 watts when increasing the drive pot before the output got dirty, and the collector choke in the PA stage became pretty warm. After an earlier posting, several great guys on this list offered some suggestions. While some of them helped a little, the instability was still there when approaching 4.5 to 5 watts.

I was able to get up to the threshold of 5 watts by using a selected 2N3904 for Q4, and a selected PN2222A for Q5. Along with the other small mods that follow, fairly stable operation at ~5 watts was possible. But if the antenna was reactive at all, or when not using a dummy load for testing puposes,or if I went even a smidge above 4.95 watts - the output spectrum began to deteriorate, and parasitic oscillations in the rig popped the output suddenly to 7-9 watts of crud. I had some limited success with higher bias on Q4 / Q5, and by reducing the lead lengths of these two transistors, but it basically wanted to "take off" on it own right before 5 watts occured on the wattmeter. Heck.

Upon very close scrutiny, and some help from Dave Benson, I found that Miller effects were to blame, and that the succeeding stages after Q4 were not directly the cause of the instability and spurious products. Rather, the source of the instability was Q4 itself. If ground loops, or RF feedback, or poor decoupling at higher power levels are the source for this instability - I can't be sure. As with the Norcal 38 Special, a lot of speculation is in effect here, but no single true solution was evident.

Onward I searched for a solution that would not require a lot of modifications to the circuit design or the PCB. I began to study some of the other designs of the rig's I own. In the S&S Tac-1, gain in the transmitter driver stages is distributed across 4 stages of amplification. One amplifier follows the NE602 mixer, and three more stages of amplification follow the TX bandpass filter before the final PA stage. This way, no single stage or pair of driver stages needs to have a lot of power gain and gain instability is avoided without such high amplification. Total TX driver gain is spread across more devices and the result is good stability and linearity.

In the Norcal Sierra and NC40A, a post-mixer FET amplifier is used to maintain high unloaded Q of the bandpass filter(s), yet adequate power gain is achieved with a single following driver stage without the loss of stability. I thought perhaps substitution of a FET transistor for the post-bandpass filter stage in the SW30+ might be worth a look.

It turns out that by using a J310 in place of Q4, normally a 2N4401 transistor, better stability was possible when running higher levels of drive to the PA stage. In fact, this was a two-fold improvement, since the FET permits much higher input impedance to appear at the output of bandpass filter T3 in the SW30+, maintaing higher loaded Q, and that allows a greater amount of low-level RF to be present at the input of the new FET predriver stage, Q4.

At first the higher input impedance possible as a result of using a FET instead of a bipolar device might seem to be the last thing you would want in an driver stage that was already unstable at high PA output levels. But the N-channel FET is inherently more stable, since it can be used at zero bias versus the positive bias needed for the bipolar stage. And, Miller effects tend to be less with a junction FET, since the inter-electrode capacitances are typically lower, and there are fewer biasing resistors required for the FET stage. At least that is how I understand it after do-

ing some reseach of Miller effect. Somebody correct me if I have it wrong.

For whatever reason, the FET allowed much greater PA output, with absolutely no instability detected by me at any time during my experiments. No combination of power and / or devices following the FET ever caused any instability in my SW30+. It is indeed rock-stable at any and all power levels and antenna loads. Heck, even with no antenna at all attached to the PA stage, I never have seen any hash on the output signal when viewed on the monitor scope. I like it.

So for those of you who want more power from your SW30+, or if your rig isn't as stable at various antenna loads and power levels as you think it should (could) be, read on.

Note: No traces are cut; some rather simple part sustitutions are made; and one less resistor is needed! These are the kind of mods I like to perform.

As in my SW40+, several incremental small gains in power output are achieved in the following manner:

MODIFICATIONS

1. Replace D13 (1N4001) with a 1N5819 or 1N5820 diode. This reduces the DC supply voltage drop. Remove the diode altogether and use a jumper in its place if you use some other form of polarity protection, such as a series fuse and reverse-biased diode from B+ to ground. Just be sure to use something to protect against reverse polarity and short circuits.

2. Replace Q3 (2N3906) with a good 2N4403 PNP or NTE159. Either of these transistors has much lower saturated ON voltage drop, and will help increase the supply voltage to Q4 and Q5. If your keying with the stock 2N3906 is too soft, or has a "tail" on it as someone recently mentioned, this change may correct that problem as well. Note that C110 (3.3uf) may need to be increased in value, since the keyed supply to U5, Q4, and Q5 is now switched "harder" and probably faster as well. I had to do this on my SW40+, and the SW30+ definitely needed a bigger cap for C110. I used a 4.7uF 16V tantalum cap for both rigs.

3. D12 (a 33V zener) has to be changed to a 36V (NTE5085A) or even a 43V (NTE5087A) zener to prevent it from shunting the higher output voltage of the PA stage, and overheating itself into destruction. Besides, since higher output is desired, we certainly don't want some of that power dissipated as heat in the protective zener!

4. As power levels approach 3-4 watts, L2 in the collector lead of Q6 starts to saturate and get warm. At 5 watts, it gets downright HOT. This power should be going to the antenna instead of heating the collector choke. I pulled out the six turn toroid and measured its value at 15 uH on an AADE L/C II meter. A careful inspection of the junk box turned up a cylindrical ferrite RF choke of the same value. I mounted it standing vertically in place of the original L2 toroid choke. No further heating was noticed, and the output also increased. If you also do this, be sure to get an RF choke that has low DC resistance and can handle the required current of the PA stage. The one I used was not epoxy coated, and was about ½ inch long and had decent sized wire windings over a iron core. It was of "unknown" origin, but possibly came out of some old Heath gear.

5. The turns on L4 were increased by one turn from 15 to 16 turns. This improved the load matching of the low-pass filter to the PA stage and further increased the output. The spectral purity does not seem to degrade - I noticed no increase in SWR on my resonant 30m dipole. The monitor scope waveform also looks nice, but it will take someone with a spectrum analyzer to determine this concretely. Check the other

componets in your low-pass filter if the output is still low. Some ceramic disc caps I tried in place of C38 got warm and reduced output. The little monolythic cap Dave supplies performs well - better than several ceramic disks I tried. Surprising!

6. R23 (22k) is removed. It is no longer required.

7. R22 (10k) is replaced with a 1M ¼ watt resistor. This just keeps the gate of the FET at zero bias with no signal. Without it, the FET's conduction drifts all over the place at low drive settings, and if the original 10k is left in, the input impedance is un-necessarily reduced and will lower the Q of the bandpass filter. The resulting filter bandwidth is more than adequate for the small 35 khz of tuning range on 30m.

8. Q4 is replaced with a J310 FET. A J309 works almost as well if that's all you can find. A MPF102 was not adequate in drive. Other N-channel junction FET's of sufficient transconductance and current rating may also work. The pinout of the PCB is not correct for the FET, therefore some lead bending or re-orientation of the FET package is required for proper connection. If you turn the flat of the J310 so that it is facing Q5, the leads can go into the PCB pads in a similar fashion to the original 2N4401. The gate goes where the base used to go, the drain goes to the collector pad, and the source goes to the emitter pad of the original EBC pad layout. Check the lead identification of your FET if in doubt. It's pretty hard to blow up the FET even if you put it in wrong, but it probably can be done! ;-)

9. Get a small TO-220 heat sink for Q6. With 5 watts of output, it has to be heatsinked. Dave Benson mentions a suitable h/s in the manual. The typical TO-220 h/s commonly available is much too large, unecessary for this power level, and it won't fit into the available space. It may be possible to bend the 2SC2078 over a little to make something else fit, but I would get the correct fiting h/s if I were you. Or make up your own as shown in the current QRPp, as Paul Harden illustrated. His artwork is too cool !

10. Check your work; attach a 50 ohm dummy and QRP wattmeter; then apply power and key the rig. Set the drive pot for <1 watt of output, and carefully peak T2 and T3 in the bandpass filter. Reduce drive as necessary to stay below 1 watt, since this protects the PA from overheating and also seems to indicate the best peaking. Repeat this alignment several times until no further peaking is obtainable. Be sure to use a proper tool so as to not crack the slug in T2 and T3.

11. The output power and the RF envelope should be clean and pure from zero output to near 5 watts or more. You will note the drive pot has greater useful range, indicating the FET is performing very well, and it is more sensitive to the low level RF voltage coming out of the TX bandpass filter. Cool!

12. If your rig still seems anemic, and is short of 5 watts output at 13.8 to 14 volts DC supply, here are some suggestions:

Remove Q5 (2N4401) and try several 2N3904, PN2222A, MPS2222A (available from Radio Shack, pn. 276-2009 , $0.59 each). One of these devices should get you that extra 200-300 mW you may need to achieve 5 Watts. I wound up with a NTE123AP as the preferred device in my rig, but several MPS2222A and 2N3904 devices were nearly as good. Nearly 6 watts is possible with my rig.

13. If you are looking for less receive current demand, install a LMC662CN CMOS op-amp for U4 as recently discussed and offered as a group purchase on the QRP-L. A savings of 6 or 7 milliamps has been reported with this mod. Be sure to wire the stereo headphone jack with the left and right channels in series to make better use

of the CMOS op-amp's performance. Enlarge the hole in the aluminum back panel for the headphone jack to clear the plastic shoulder of the stereo 3.5mm jack, then use a flat fiber washer under the knurled nut to insulate the metal jack from the back panel.

SUMMARY

On my rig, I can get a nice solid 5 watts output with the drive pot less than full on. I have never seen any spurious output at any drive level or with any antenna load condition. The rig is very stable at all power levels and loads. Keying is great, and the transmitter behaves as you really want it to. Even if you don't run it at the full output possible, it is nice to know the potential is there (operation on battery power can now have more possible output) and the stability and alignment is certainly better than before these mods. If you have any questions or feedback on these mods, please let me know by email. Thanks to Dave Benson, Glen Leinweber, and others for their suggestions. Their inputs helped me to zero in on the solution I was looking for. Thanks guys!

This 30m SWL rig was the result of a door prize at the Ft. Tuthill hamfest in July, and I would like to thank everyone who made that possible, especially Dave Benson who graciously made it possible and offered to swap the 40m kit I won for the 30m kit that is the subject of this article. I, of course, had already purchased a 40m kit earlier in the Elmer101 project.

PS. One change I still need to make to the SW40+, is to replace the collector choke on it, since it also heats at power levels approaching 5 watts, though not as much as the SW30+ did. ;-)

72 and good luck, Gary Surrency AB7MY
Chandler, AZ (near Phoenix)

A Homebrew Enclosure for your Elmer 101 Rig

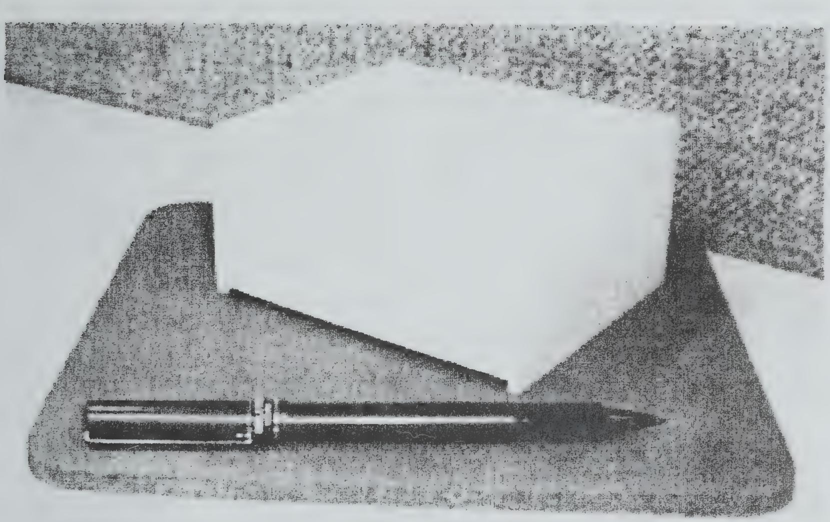
by Bill Jones, KD7S
83Redwood Ave.
Sanger, CA 93657

This is a series of step-by-step instructions on how to build custom cabinets for popular board kits. This project is designed around the Small Wonder Labs SW+ transceiver made popular by the qrp-l Elmer 101 tutorial series.

This cabinet is unique in that it requires no fasteners to hold it together.
Instead, it use a "tongue and groove" design. That is, the base of the enclosure slides into grooves cut in the sides. Likewise, the front and rear panels are seated in grooves cut in the sides and base. The pieces lock together to form a rock solid, "hardwareless" enclosure. Trying to remove a rusted screw from a conventional box with a butter knife may be a thing of the past.

If you haven't already done so, please read the section, Learn how to make homebrew enclosures. Here is where you will find the basics on how to layout, cut, weld and finish ABS plastic.

Some fine print. Just so there is no misunderstanding, I received written permission from Dave Benson, NN1G, to target the SW+ transceiver. You may already know that Dave sells a matching hardware package which includes cabinet. However, according to Dave the hardware pack is offered mainly as a courtesy to the builder. If you elect to build an enclosure from the design information presented here, please be careful. I recommend the use of a table saw to cut the plastic and a highly flammable liquid as a bonding agent. I cannot be responsible for any accidents resulting from their use.



hereby authorized to duplicate it for personal use only. That means you don't have the right to make a dozen cabinets and sell them at the next ham club meeting. If you do, you are breaking the law and things like that make me really cranky.

Be your own Cabinet Maker

How to build custom electronic enclosures from scrap plastic:

Anybody who has built a piece of electronic equipment has faced the problem of finding a suitable enclosure. Commercially made boxes never seem to be the right size, assuming you can find one in the first place. Even if you do locate what you need, prices are often outrageous.

If you think this is just another problem you have to live with, think again. With a few square inches of sheet plastic and some time at your workbench you can roll your own custom cabinets. You can build them any shape or size you need using common shop tools. The best part is that the finished product will rival commercial enclosures in both looks and functionality, but at a fraction of the cost.

The two-piece, clamshell enclosure shown above was custom made to house a Small Wonders Labs SW40+ transceiver kit. The top and bottom pieces are held together with a pair of "NorCal 40" plastic latches. The front and rear panels slide into grooves cut in the side pieces and are recessed 1/8 inch. All that remains is to drill the end panels to accommodate the controls and connectors and add some stick-on rubber feet. The end result is a rugged, attractive homebrew cabinet that cost less than $2.00 to build.

Follow the steps below for instructions on how to make your own enclosures.

ABS plastic - The "Volkswagen" of cabinet making.

The heart of homebrew enclosures is a type of rigid thermoplastic known as ABS (Acrylonitrile-Butadiene-Styrene). ABS can be sawed, drilled, sanded, machined and painted. When heated to around 250 degrees Fahrenheit, ABS can be bent or even vacuum-formed. It is rugged, light weight and cheap. ABS is available in several different colors including white, black, beige and gray. Scraps of ABS can be found at plastic supply houses or sign shops for a dollar or two per pound. A pound of 1/8" sheet will make a lot of small

cabinets. Check your telephone book's yellow pages for suppliers.

Other types of sheet plastic can also be used. An article describing the use of acrylics and polycarbonates can be found in the August, 1988 issue of QST magazine.

Drawing a design and cutting the pieces to size

In its simplest form an electronics enclosure may consist of nothing more than a few pieces of plastic, welded and screwed together, to form a box.

The best way to learn about ABS is to start with a basic enclosure. Begin by drawing the outline for the top, bottom, sides and end panels on a sheet of paper. Use a sharp pencil or fine line permanent marker and a steel ruler. A T-square or carpenter's framing square will help considerably. When the layout and dimensions are correct, transfer the pattern to some plastic stock. Make sure all lines are parallel. If not, your project will not be square.

Cutting the panels is best done with a tablesaw fitted with a fine toothed blade. If you don't have a table saw, don't dismay. A hacksaw or other type of finishing saw will work just fine. It will just take longer, that's all. Whatever tool you use, cut slowly and make sure your blade doesn't wander back and forth across the line.

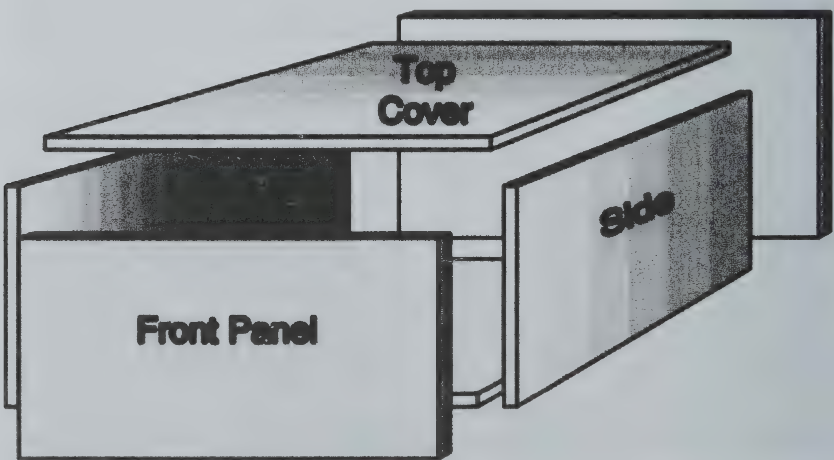
If the saw blade gums up in the plastic, change the blade speed. It doesn't take much friction to heat the plastic to its melting point. When you find the right setting you can slice through sheet plastic just like the professionals.

Welding the Pieces Together

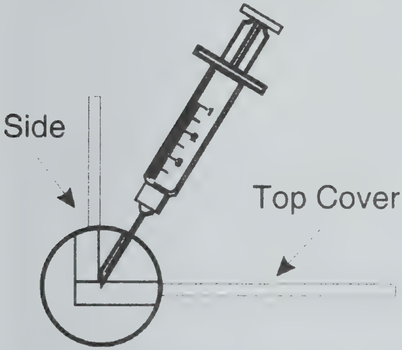
With wood you glue - with plastic you weld

Once the pieces are cut to size you're ready to weld the box together. You can weld ABS using common PCV pipe cement from your local hardware store. If you want to get fancy, there are special, water-thin bonding agents available from plastic supply houses. While you're there you can pick up a special syringe dispenser that makes it much easier and neater to apply. Far from being just another type of glue, these bonding agents actually melt the plastic pieces to be joined. When the solvents evaporate you end up with a chemically bonded joint that is almost as strong as the original material.

WARNING! Be absolutely sure you have adequate ventilation while working with these chemicals. They are extremely flamable and can be harmful if inhaled.



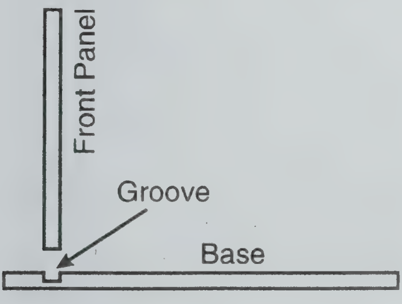
Assembly begins by welding the two sides, one at a time, to the top. Coat both surfaces to be joined with a light application of solvent. A toothpick makes a good application tool. Immediately press the pieces together and apply moderate pressure for one or two minutes. Check for



proper alignment—make sure the parts are square. Set everything aside for an hour or so to allow the the solvents to evaporate. Repeat this step for the other side.

If you're using a syringe and the liquid solvent, simply press the pieces together (clamp them in place if you can) and run a bead of solvent between the parts to be joined as shown in the drawing. The liquid will immediately be drawn into the joint area by capillary action.

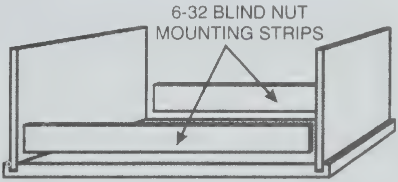
Next, join the two end panels to the base. If you want to make absolutely sure the panels are aligned properly, use your table saw to cut a pair of shallow grooves in the base. Then slip the panels into the



grooves before welding. Again, if you don't have a tablesaw, make a series of shallow, parallel cuts with a hacksaw. Then use the edge of a file to even up the cuts to a uniform depth.

Keep in mind that it is much easier to drill holes in these panels for controls, connectors, meters, dials and such before they are welded into place.

Next, cut and weld two rectangular strips of scrap to the base. These will be used to mount four 6-32 blind nuts which will hold the cover in place. Give the solvent plenty of time to evaporate so as to form a strong bond. Then temporarily position the cover over the base and drill four pilot holes through both the cover and the rectangular mounting strips. Next, apply



a thin coat of solvent around the insides of the holes in the strips and press the blind nuts into place. Use a pair of pliers to embed the tabs into the plastic for added strength.

All that remains is to add any finishing touches such as paint or decals. See the next section for a discussion on sanding and painting.

Sanding and painting

If it doesn't look quite perfect, here's how to fix it.

You will probably discover a few rough edges, nicks and scratches or other anomalies that need to be smoothed out. ABS sands very nicely so it is a simple task to erase any slight imperfections.

Tape a whole sheet of medium grade sandpaper to a flat, smooth surface. Run the assembled box back and forth over the

sandpaper using light pressure. Turn the box in your hand from time to time to make sure you don't remove too much material from any given area. Repeat the process with a finer grade of sandpaper. You can finish the job by hand rubbing the enclosure with some fine-mesh steel wool.

ABS will accept spray paint very well. A coat or two of primer will help the paint adhere better. Read and follow the directions on the paint can for best results. And, as always, two light coats of paint are better than one heavy coat.

Building the Elmer 101 Enclosure

Measuring and cutting the pieces

The following instructions will show you, one step at a time, how to cut and prepare the six pieces of ABS plastic that go into making up the Elmer 101 cabinet. Please note that a table saw fitted with a plywood or plastic cutting blade is almost essential.

As each piece is cut, label it with a piece of masking tape as shown in the photograph in Fig. 2. All pieces are cut from 1/8" thick sheet

Instructions

1. Cut a piece 4 3/8" by 3 7/8" and label it TOP.

2. Cut another piece 4 1/4" by 3 7/8" and label it BASE.

3. Cut two pieces 2 1/8" by 3 7/8" and label them LEFT END and RIGHT END respectively.

4. Cut two more pieces 4 1/4" by 1 15/16" and label them FRONT PANEL and REAR PANEL respectively.

In the following steps you will cut grooves in the LEFT and RIGHT ENDS and the BASE.

Raise the table so that the top of the blade is exactly 1/16" above the table surface. Using a piece of scrap, make a few test cuts to insure the groove depth is precisely half the thickness of your sheet plastic stock. Next, position your rip fence so that the outer edge of the kerf is 1/8" away from the edge of the stock being cut.

There will be a total of eight grooves to cut—three in the RIGHT END, three in the LEFT END and two in the BASE. It will probably be necessary to move the rip fence slightly (after the initial cuts have been made) in order to widen the cuts to a uniform 1/8" width. Just be sure to make all eight cuts before you move the fence.

5. Select the BASE to make the first cuts. Position the long edge (4 1/4") against the rip fence and cut the first groove. Make a couple passes to insure a clean, smooth cut. Then, turn the BASE around and make a similar cut on the opposite side.

6. Select one of the ENDS. Place the long edge (3 7/8") against the rip fence and cut a groove. As with the BASE, make a couple passes. Next, place the short edge (2 1/8") against the rip fence and make a cut. Finally, make a third cut on the opposite end.

7. Repeat step 6 using the other END.

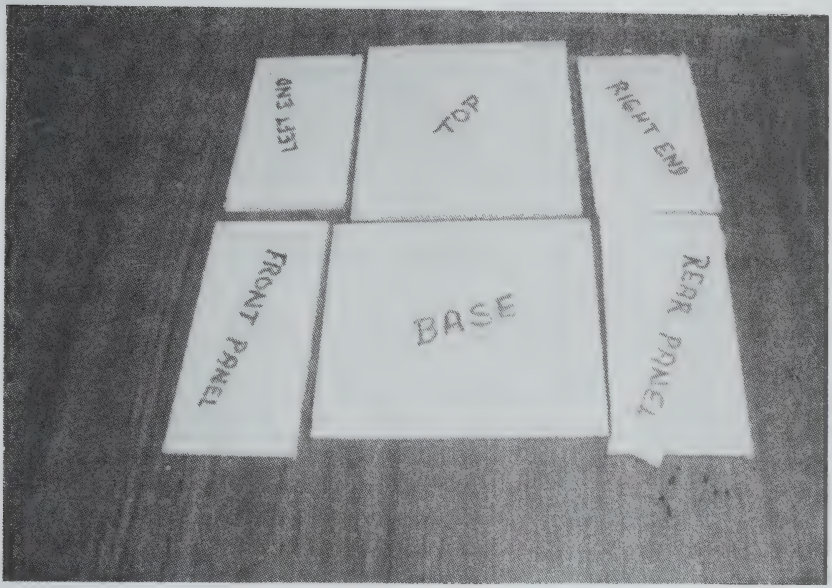
8. Now, using a piece of scrap, make a test cut. Then move the rip fence away from the blade very slightly and make a second pass. Take another piece of scrap and see if the edge will fit (snugly) into the groove you just cut. If it is too narrow, move the rip fence again and make another cut. Keep this up until you have a groove that is a tight fit with the edge of your sheet plastic stock.

9. Go back over all eight grooves with the current settings of your rip fence.

10. Use your fingernail to remove any burrs or rough edges. You're now ready to proceed to the assembly step.

Assembly - Getting it together, at last

The hard part (cutting the pieces to size and forming the grooves) is behind you. All you need do now is weld the RIGHT END and LEFT END to the TOP. Still, don't get in a hurry. Do the job right and you will be very pleased with your

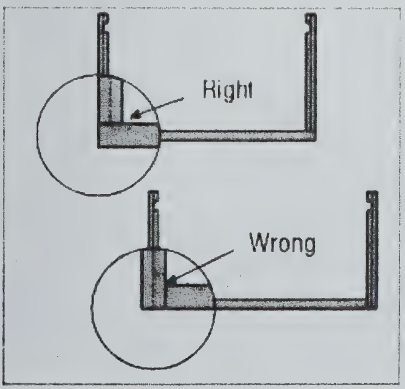


handiwork.

1. Select the cabinet TOP and remove any burrs from the edges.

2. Repeat step 1 with the LEFT and RIGHT ENDS. You want to make sure the pieces are smooth and clean.

In the following steps you will weld the LEFT and RIGHT ENDS to the TOP. Make absolutely sure that you put the END on top of the TOP, not beside it. See the illustration at the bottom of the page if you're confused.

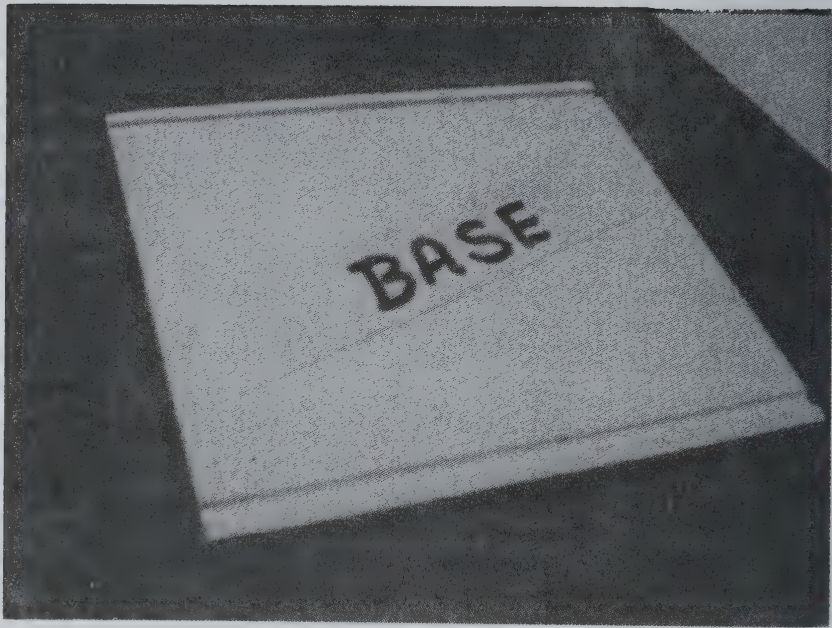


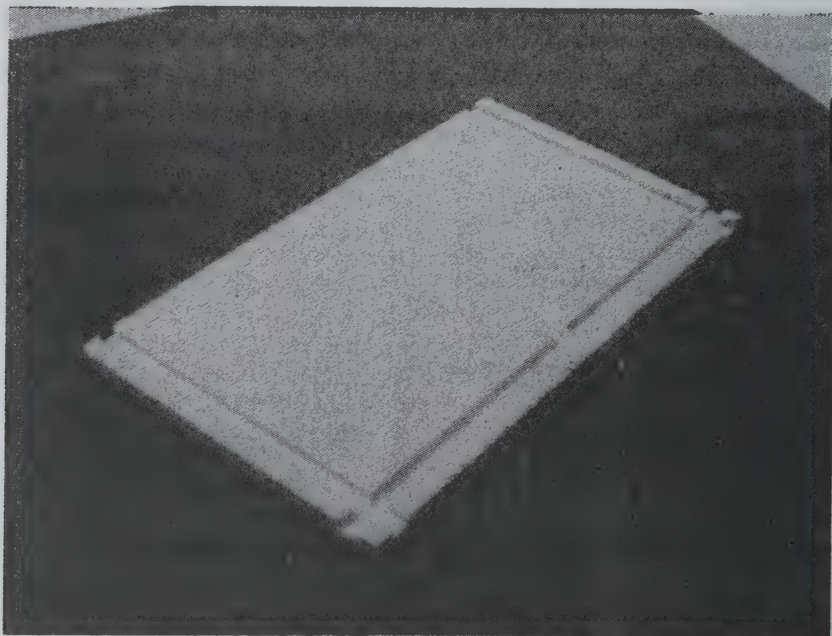
Keep in mind that you will be welding the pieces one at a time. Allow the first joint to cure completely before attaching the opposite END. Depending upon the type of solvent you use, this shouldn't take more than a half hour or so.

IMPORTANT:

Check the fit of the END panels to the TOP before applying solvent. Make sure they will fit together tightly. Also, when you do apply the solvent, angle the free ends of the ENDS toward the center of the cabinet very slightly. This will put tension on the BASE, FRONT and REAR PANELS when the completed box is snapped together.

Weld each END, one at a time, to the top. Apply moderate pressure to the parts being joined for at least 30 seconds. Confirm that the edges match up properly and there is no offset or misalignment except


BASE



for a slight "lean" toward the center. After both joints have cured, apply a light fillet of solvent to the inside of each joint to fill in any possible gaps. Set the assembly aside for a couple hours to cure. You will most likely have a few rough spots on the outside where the ENDS were joined to the TOP. These can be removed with the edge of a metal ruler used as a scraper.

Finish Work

Cleaning up the loose ends

All that remains is to check the FRONT and REAR PANELS and the BASE for proper fit. You will probably have to touch them up slightly.

1. Insert the BASE into the grooves in LEFT and RIGHT ENDs. The ENDs of the box should be parallel with one another. If not, the BASE may have to be trimmed slightly. Successive passes through your table saw should do the job. Just make sure you don't take off too much material.

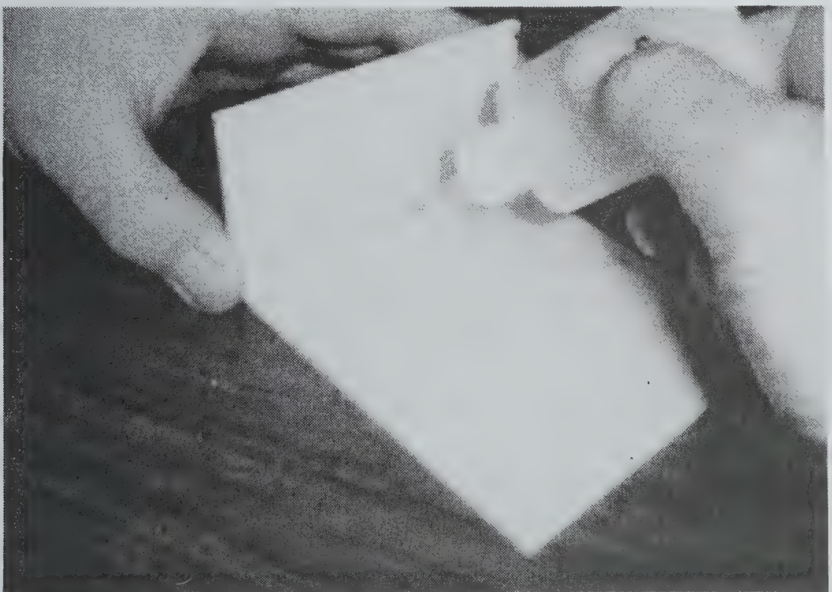
2. Check the FRONT and REAR PANELS for fit. It is likely they will not slide far enough up to make contact with the TOP. If that's the case, round the corners of the FRONT and REAR PANELS slightly by dragging them across a sheet of medium grit sandpaper taped to a flat surface.

3. Once all the pieces fit together smoothly, assemble the cabinet by sliding the FRONT and REAR PANELS into place. Then spread the LEFT and RIGHT ENDs just enough to accept the BASE. Push everything together so the parts lock into place in the grooves. Hey! It looks pretty good, doesn't it?

You can remove any imperfections (like saw marks) from the edges of the panels by running the assembled enclosure over a sheet of fine grit sandpaper taped to a flat surface. You may want to round the corners where the ENDs are attached to the TOP with a sanding block while you're at. Finish the cabinet by attaching some stick-on rubber feet to the BASE.

Some Additional Thoughts

There are some excellent computer based drawing programs available that can make it easy to generate custom artwork for your homebrew projects. Likewise,



color printers seem to be everywhere nowadays.

For a small investment in time you can create professional looking paper overlays that transform your homebrew projects into a true work of art.

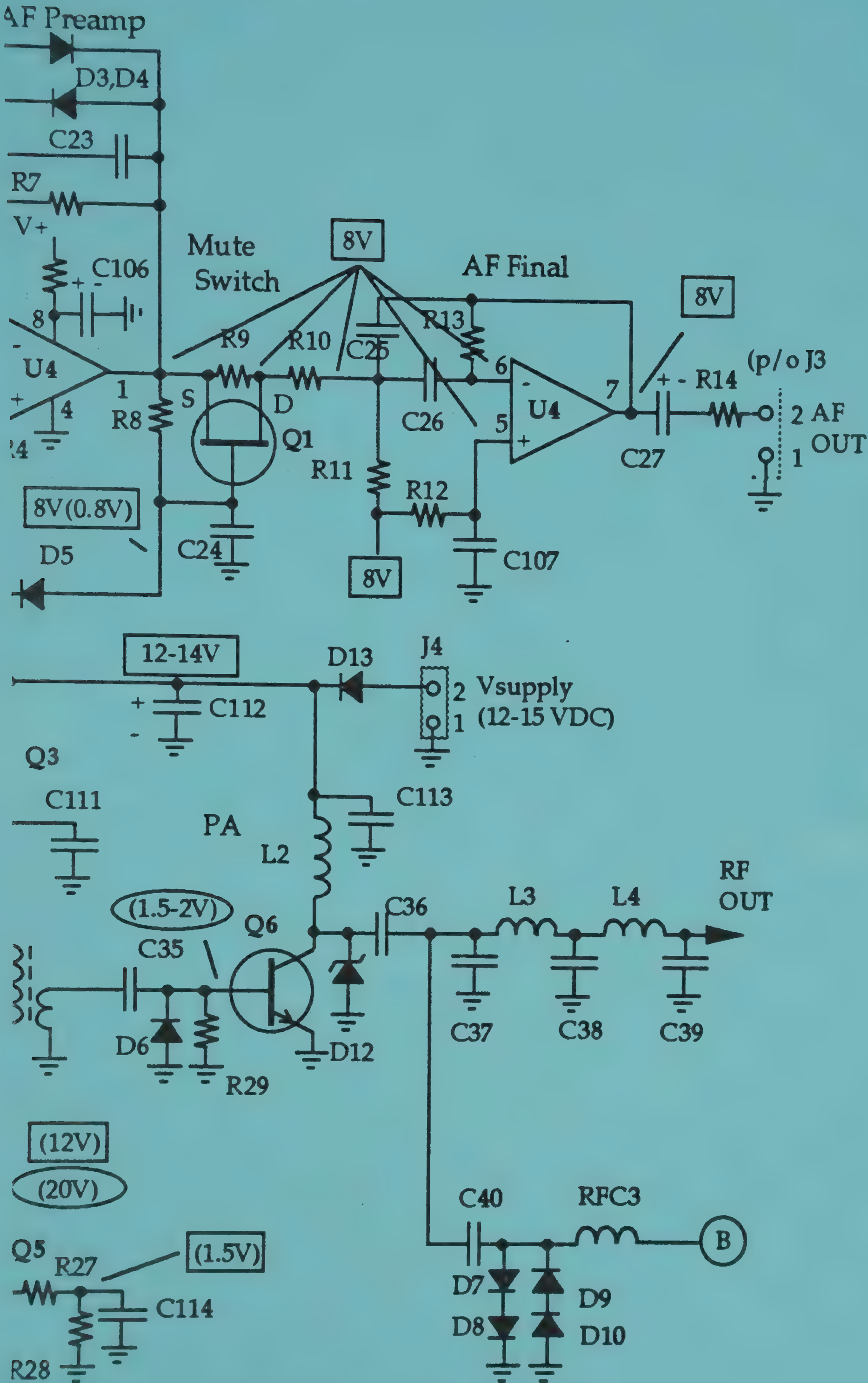
Imagine the reaction you will get from your fellow hams when you show them your homebrew transceiver, housed in a homebrew cabinet, with homebrew panel markings. You're gonna' love it.

NorCal QRP Club Web Page

Jerry Parker, WA0OWR
Webmeister

http://www.fix.net/norcal.html

The place to go on the web for the latest up-to-date information on NorCal. Jerry has monthly reports of NorCal meetings, complete with pictures, details on the latest club project, articles of interest to QRPers, and information on how to order an official NorCal T-Shirt.



oting the "SW-XX+" Transceiver

IDER LABS™

)/98

SCHEMATICS

color printers seem to be everywhere nowadays.

For a small investment in time you can create professional looking paper overlays that transform your homebrew projects into a true work of art.

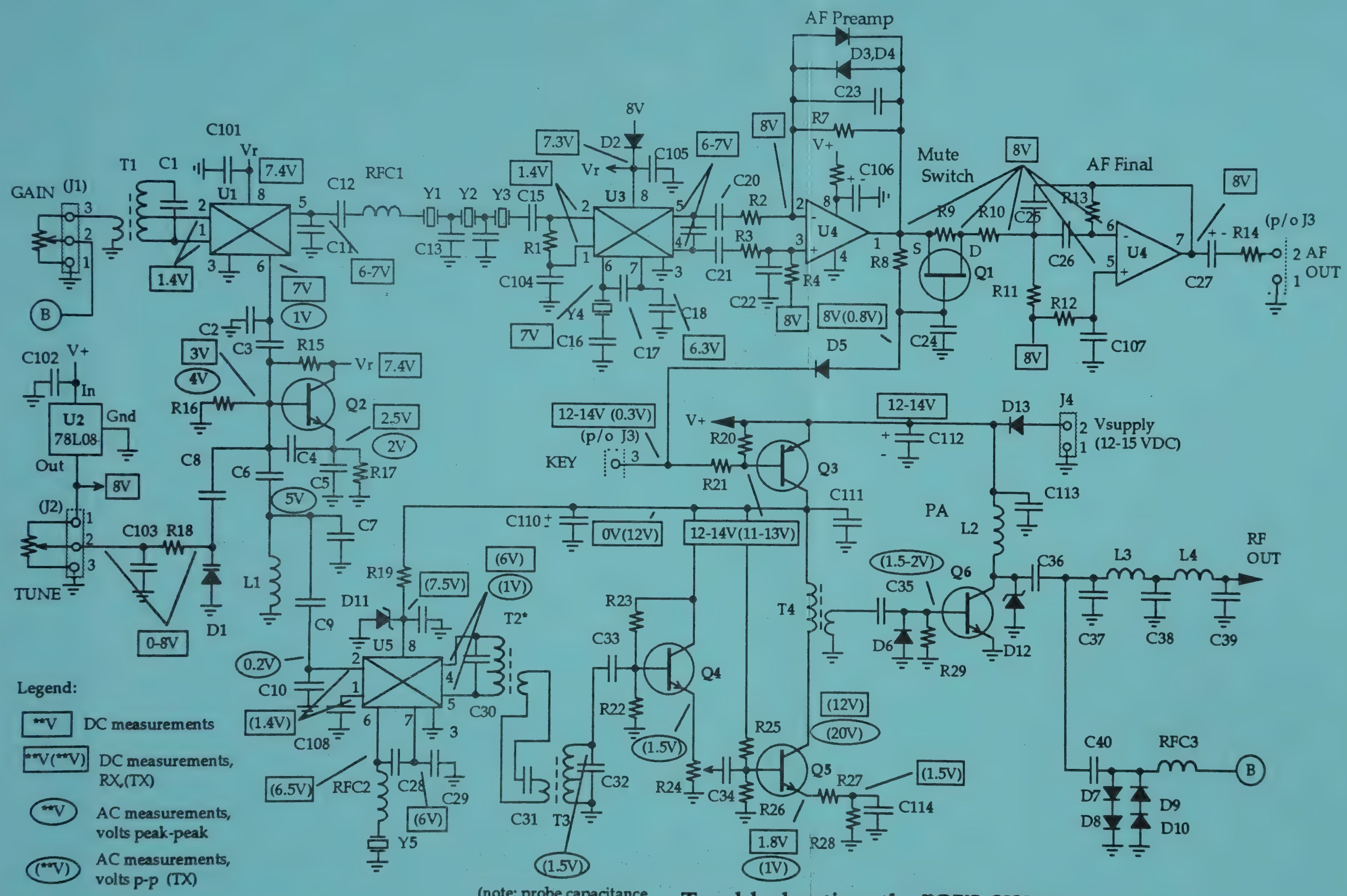
Imagine the reaction you will get from your fellow hams when you show them your homebrew transceiver, housed in a homebrew cabinet, with homebrew panel markings. You're gonna' love it.

NorCal QRP Club Web Page

Jerry Parker, WA0OWR
Webmeister

http://www.fix.net/norcal.html

The place to go on the web for the latest up-to-date information on NorCal. Jerry has monthly reports of NorCal meetings, complete with pictures, details on the latest club project, articles of interest to QRPers, and information on how to order an official NorCal T-Shirt.



Legend:

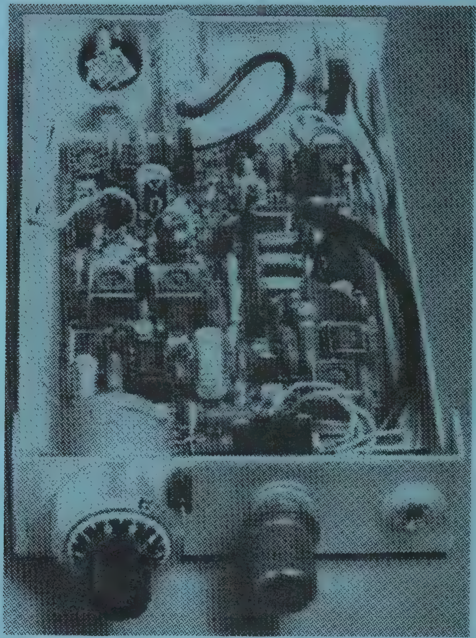
**V	DC measurements
V(V)	DC measurements, RX,(TX)
**V	AC measurements, volts peak-peak
(**V)	AC measurements, volts p-p (TX)

// readings " downstream" of TX drive pot R24 are for maximum drive setting.

Troubleshooting the "SW-XX+" Transceiver

SMALL WONDER LABS™

3/10/98



Glen Leinweber's (VE3DNL) completed kit and the SW-40+ upon which his excellent test data in this issue was based.

Bruce Rattray, VE5RC/VE5QRP, checking his work following Lesson 1. Note signal generator and O-scope, helpful aids. Bruce's SW-40 is now completed and making QSO's

NorCal-20 Update

The **NorCal 20 kit** is proceeding along nicely, but like anything of this magnitude, not without a few "glitches." With around 250 parts per kit, times 1000 kits, NorCal has exhausted the stock-on-hand of several pertinent parts from our suppliers. All partial deliveries and back-orders will be fulfilled by December 15th. In the meantime, much of the kitting has begun, including matching of all the crystals now completed (a heroic effort in itself) and the sorting of over 16,000 toroids.

Kits should be completed by the end of December, with shipping of the kits to begin shortly after the New Years holiday. We don't want to start shipping the kits during the Christmas "rush." The remaining 500 kits will be shipped to Rev. George Dobbs at G-QRP about a month later for third-world distribution.

Thank you for your patience. This is the most ambitious kit ever undertaken by NorCal, and the most extensive design effort expended on a NorCal kit. You will be pleased with Dave Fifield's design and the performance of this outstanding addition to your QRP station.

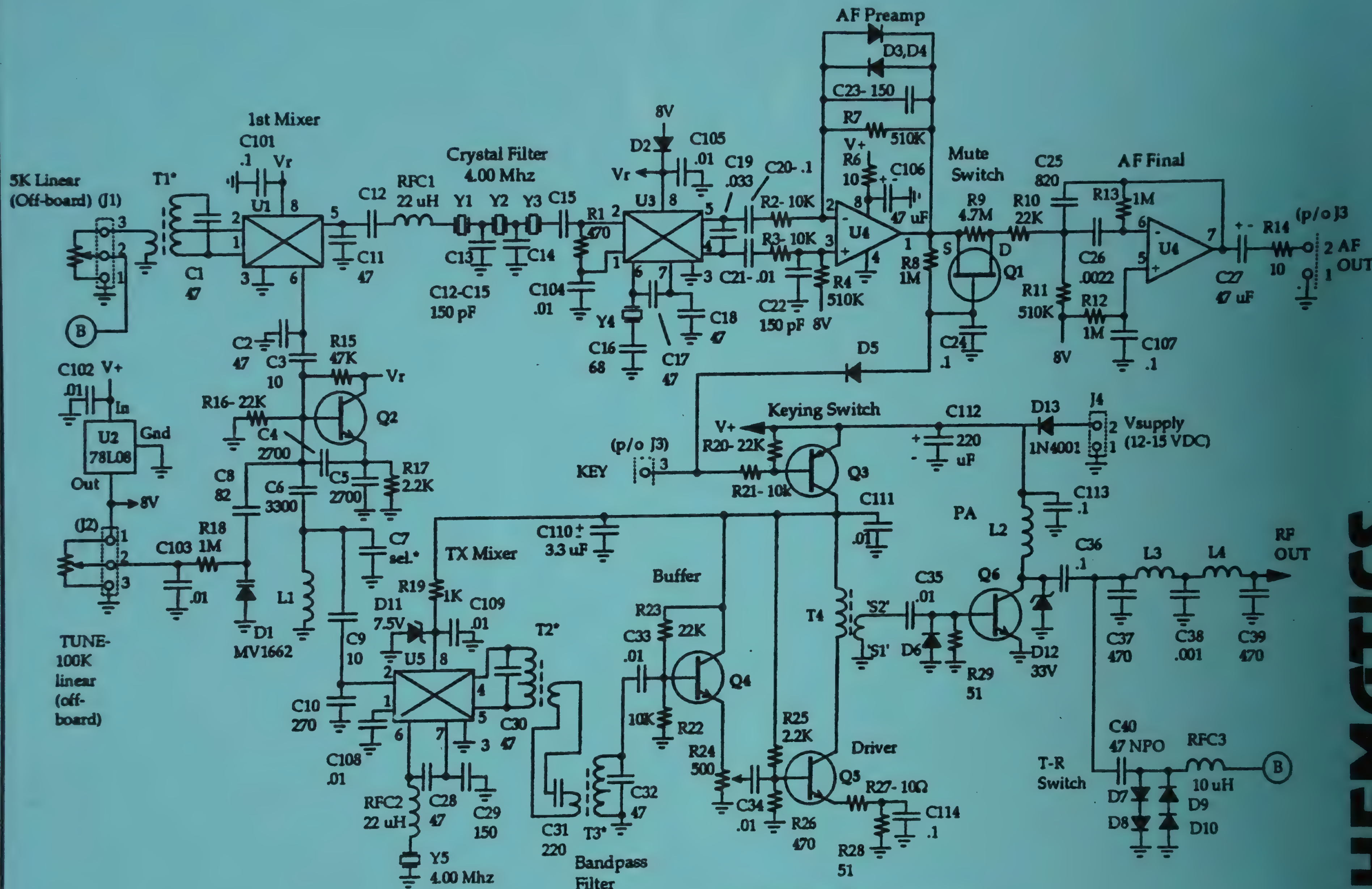
72, Doug Hendricks, KI6DS and Jim Cates, WA6GER

QRPp Production Status

We apologize that this issue of QRPp is about a month late. Production was initially delayed due to time required to prepare for this year's PacifiCon, and delayed further as we decided to change the format of this special issue to nearly 90 pages and the special binding. This required all of the pages of QRPp to be redone to meet the printing requirements for this type of bindry, the special fold-out schematics, and the time to have it performed by a professional bindry service. We hope you like the appearance of this special issue.

The good news is that the Winter QRPp is already in progress, which will feature the Jim Kortge 2N2 build-it-from-scratch rig. The Winter QRPp will ship on time, in mid-January, 1999.

72, Paul Harden, NA5N



The "SW-40+" Transceiver

SMALL WONDER LABS™

3/18/98

* T1-T3
C-internal not shown,
leave intact.

NorCal-20 Update

The **NorCal 20 kit** is proceeding along nicely, but like anything of this magnitude, not without a few "glitches." With around 250 parts per kit, times 1000 kits, NorCal has exhausted the stock-on-hand of several pertinent parts from our suppliers. All partial deliveries and back-orders will be fulfilled by December 15th. In the meantime, much of the kitting has begun, including matching of all the crystals now completed (a heroic effort in itself) and the sorting of over 16,000 toroids.

Kits should be completed by the end of December, with shipping of the kits to begin shortly after the New Years holiday. We don't want to start shipping the kits during the Christmas "rush." The remaining 500 kits will be shipped to Rev. George Dobbs at G-QRP about a month later for third-world distribution.

Thank you for your patience. This is the most ambitious kit ever undertaken by NorCal, and the most extensive design effort expended on a NorCal kit. You will be pleased with Dave Fifield's design and the performance of this outstanding addition to your QRP station.

72, Doug Hendricks, KI6DS and Jim Cates, WA6GER

QRPp Production Status

We apologize that this issue of QRPp is about a month late. Production was initially delayed due to time required to prepare for this year's PacifiCon, and delayed further as we decided to change the format of this special issue to nearly 90 pages and the special binding. This required all of the pages of QRPp to be redone to meet the printing requirements for this type of bindry, the special fold-out schematics, and the time to have it performed by a professional bindry service. We hope you like the appearance of this special issue.

The good news is that the Winter QRPp is already in progress, which will feature the Jim Kortge 2N2 build-it-from-scratch rig. The Winter QRPp will ship on time, in mid-January, 1999.

72, Paul Harden, NA5N

QRPp, Journal of the NorCal QRP Club
862 Frank Ave.
Dos Palos, CA 93620

PRESORT
FIRST-CLASS
Permit #72
Socorro, NM
87801

SOC 127

Phil Westover
Summer 00
10340 SW Hillview St.
Tigard OR 97223

Pre-sort
FIRST CLASS

NorCal's T-Shirts

NorCal offers two shirts:The NorCal T-Shirt and The NorCal Zombie T-Shirt (Limited Edition) The price is $15 each plus $3 shipping and handling in the US, $5 shipping for DX. The shirts are the recognizable NorCal "GOLD" and high quality and heavy duty. The NorCal shirt is imprinted with the NorCal logo and the NorCal Zombie shirt is imprinted with the NorCal Zombie 'Toon. The shirts are gold with the NorCal Logos in black and the Zombie 'Toon is multicolor. To order Send $15 + $3 postage ($5 DX) to: Jerry Parker,
426 Tanglewood Ct.
Paso Robles, CA 93446

Don't forget to specify your size: M, L, XL, XXL (Note XXL shirts are $3 additional) Please make check or money order out to Jerry Parker, NOT NORCAL, US Funds Only.

QRPp Back Issues Pricing:

1993 - $10, 1994 - $15, 1995 - $15, 1996 - $15, 1997 - $15 (Avail. Feb. 1, 1998) Full year sets available. NO individual issues available, sets will not be broken up.

Shipping: US

$3 for 1 - 3 issues, $5 for 4 - 5 issues.

Shipping: Canada

$3 for 1 issue, $5 for 2 - 3 issues, $7 for 4 -5 issues.

Shipping: DX Europe & South America

$5 for 1 issue, $7 for 2 - 3 issues, $10 for 4 - 5 issues

Shipping: DX Pacific Rim, Australia & New Zealand

$5 for 1 issue, $10 for 2 issues, $15 for 3 issues, $20 for 4 issues, $25 for 5 issues

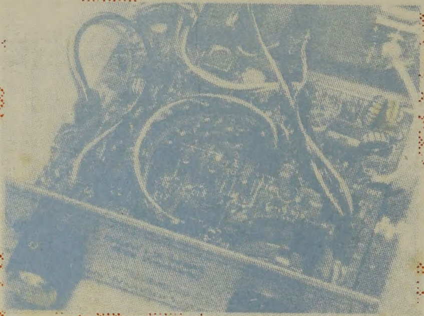
All funds US funds only. Make check or money order to Doug Hendricks, NOT NorCal. Please send orders to: Doug Hendricks , 862 Frank Ave. , Dos Palos, CA 93620, USA

QRP Frequency Crystals

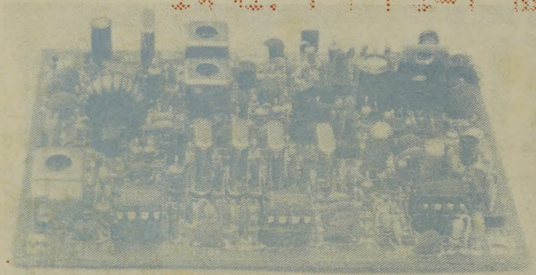
NorCal has available the following crystals in HC49U cases for $3 each postage paid in th e following frequencies: 7.040 MHz, 7.122 MHz., 10.116 MHz. Send your order and payment in US Funds only to: Doug Hendricks , 862 Frank Ave. , Dos Palos, CA 93620, USA. Make check or money order to Doug Hendricks, NOT NorCal.

QRPp Subscriptions

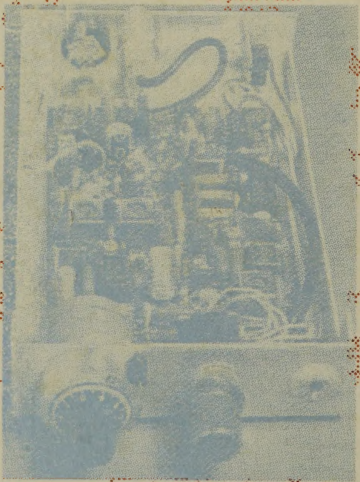
QRPp is printed 4 times per year with Spring, Summer, Fall and Winter issues. The cost of subscriptions is as follows: US and Canadian addresses: $15 per year, issues sent first class mail. All DX subscriptions are $20 per year, issues sent via air mail. To subscribe send your check or money order made out to Jim Cates, NOT NorCal to: Jim Cates, 3241 Eastwood Rd., Sacramento, CA 95821. US Funds only. Subscriptions will start with the first available issue and will not be taken for more than 2 years. Membership in NorCal is free. The subscription fee is only for the journal, QRPp. Note that all articles in QRPp are copyrighted and may not be reprinted in any form without permission of the author. Permission is granted for non-profit club publications of a a non-commercial nature to reprint articles as long as the author and QRPp are given proper credit. The articles have not been tested and no guarantee of success is implied. If you build circuits from QRPp, you should use safe practices and know that you assume all risks.



Front and rear panel views of the SW 30+ built into a Ten-Tec enclosure by Adam Kanis, N2BRT, Iowa City, IA.
http://genome33.ped-gen.uiowa.edu/hamradio/SW30+.html



The original SW 40+ built by the designer, Dave Benson, NN1G, Newington, CT



The ultimate "smoke test" -- Bruce Rattray, VE5RC/VE5QRP, Regina, Saskatchewan, making his first QSO with the SW 40+ he built

The SW 40+ built into a custom enclosure by Glen Leinweber, VE3DNL